"These authors are a virtual 'dream team' of Revit Structure expertise."
—From the Foreword by Nicolas Mangon, Senior Structural Business Line Manager, Autodesk, Inc.

Thomas S. Weir
Eric Wing
Jamie D. Richardson
David J. Harrington

MASTERING Revit® Structure 2009

Harness the Power of BIM in Your Structural Engineering Projects

Master Key Techniques and Improve Your Productivity

Autodesk® Authorized Publisher
Autodesk® Authorized Author
SYBEX® SERIOUS SKILLS.

Mastering

Revit® Structure 2009

Mastering

Revit® Structure 2009

Thomas S. Weir

Jamie D. Richardson

Eric Wing

David J. Harrington

WILEY

Wiley Publishing, Inc.

Senior Acquisitions Editor: Willem Knibbe
Development Editor: Thomas Cirtin
Technical Editor: David J. Harrington
Production Editor: Melissa Lopez
Copy Editors: Elizabeth Welch and Linda S. Recktenwald
Production Manager: Tim Tate
Vice President and Executive Group Publisher: Richard Swadley
Vice President and Executive Publisher: Joseph B. Wikert
Vice President and Publisher: Neil Edde
Compositor: Craig Johnson, Happenstance Type-O-Rama
Proofreader: Jen Larsen, Word One
Indexer: Ted Laux
Cover Designer: Ryan Sneed
Cover Image: © Pete Gardner/ Digital Vision / Getty Images

Published simultaneously in Canada

ISBN: 978-0-470-38440-4

Library of Congress Cataloging-in-Publication Data
Mastering Revit structure 2009 / Jamie Richardson ... [et al.]. -- 1st ed.
p. cm.
ISBN 978-0-470-38440-4 (paper/website)
1. Architectural drawing--Computer-aided design. 2. Architectural design--Data processing. 3. Autodesk Revit. I. Richardson, Jamie, 1975-
NA2728.M396 2009
720.28'40285536--dc22
2008040306

10 9 8 7 6 5 4 3 2 1

Dear Reader,

Thank you for choosing *Mastering Revit Structure 2009.* This book is part of a family of premium-quality Sybex books, all of which are written by outstanding authors who combine practical experience with a gift for teaching.

Sybex was founded in 1976. More than thirty years later, we're still committed to producing consistently exceptional books. With each of our titles we're working hard to set a new standard for the industry. From the paper we print on, to the authors we work with, our goal is to bring you the best books available.

I hope you see all that reflected in these pages. I'd be very interested to hear your comments and get your feedback on how we're doing. Feel free to let me know what you think about this or any other Sybex book by sending me an email at nedde@wiley.com, or if you think you've found a technical error in this book, please visit `http://sybex.custhelp.com`. Customer feedback is critical to our efforts at Sybex.

Best regards,

Neil Edde
Vice President and Publisher
Sybex, an Imprint of Wiley

To my Dad, William Weir, and my Grandfather, Clayton Sanford. I wish they were here to see how my journey has turned out.
—Thomas S. Weir

To my big brother, Shon, and my best friend, Baley, whom I think about and miss every day.
—Jamie D. Richardson

To my mom, Susie, because she never gets credit but no doubt made me who I am today.
—David J. Harrington

Acknowledgements

Thanks to my wonderful wife, Aida Gonzalez, for all the love and support she has given me this year while I worked all these odd hours writing this book. A wonderful photographer, she also took my photo! Thanks also to Sally, Dylan and Janice, and Josie and Steve. This is also for my grandson, Josh, so he knows that anything is possible if you are willing to work hard and really want it to happen. And as always I thank my mom, Estelle, for her constant and wise support.

I want to thank my colleagues David, Jamie, and Eric for their great effort over these last six months in creating this text. There was no blueprint to follow since no other books on this subject have been published, but our combined creative forces made it happen.

I want to thank all my valued colleagues at Brandow & Johnston for supporting me through this venture. I want especially to thank Gabriel Lopez and Gautam Shenoy for their talented assistance in preparing the glossy color insert pages. I also want to thank our friend Paul Anderson for his assistance in preparation of the first chapter. And thanks to my colleague Ken Gebhart for letting me use his house model.

Thanks to Nicolas Mangon for helping create the last five amazing years, and to all the dedicated, imaginative crew at Revit Structure in Waltham. They are true revolutionaries in our field.

This whole venture also has to give a nod to the people at our publisher, Wiley, who gave us the chance and provided great professional editors who have helped us to craft a first-rate text.

—Thomas S. Weir

To my wife Katie: I can't thank you enough for giving me the support I needed and for always being there to provide encouragement when it was needed. Yes, there were days early on when I didn't think I could do this. I also want to thank my daughter, Avery, and son, Layton, for understanding why Dad was always so busy. I hope that someday this book will inspire you to attempt things you didn't think were possible. Never have the "I can't do it" attitude.

Thanks to my fellow authors Tom, Eric, and David for giving me the opportunity to be part of the team. It has been great getting to know you better and working with you. A special thanks to Tom for hanging on in the beginning to help make this happen.

Thanks to Ericksen, Roed and Associates for supporting me throughout my writing efforts and our Revit Structure user base for challenging me every day as well as maintaining great attitudes. You are the ones who help keep me motivated and encourage me to learn more.

Thanks to the Revit Structure team (the Factory) for listening to their user base and continually adding functionality to each new release. There are too many names to mention, but you have all listened to my comments and have provided support when I needed it.

Thanks to the AUGI forum community and those who blog for creating such a great source of information. It is great to see so many people sharing as well as expanding their knowledge.

Thanks to the entire Wiley team; you have been great to work with. I will be the first to admit that I was a bit overwhelmed with my first round of edits. In the end they all made sense. I appreciated your comments and suggestions.

This book has been a great experience.

—Jamie D. Richardson

Of course I would like to thank my family for allowing the weeknights and weekends to become "leave Daddy alone time" and for also allowing the house to fall into disrepair as I toiled in the office. You may now cash in on all of the "Soon, I promises."

—Eric Wing

First off, thank you to the best wife a man could have. Carla has made this writing effort the best ever, with all the encouragement and support that I needed. Second, I need to thank the kids for giving me the time away to write: Kara, Ricky, Kaitlin, Kaylee, Krista, and Christopher—yes, I now have time to go into the pool. And as always I thank my dad John, who even though he hasn't been here for over 10 years is still my inspiration to write about what I know.

To my fellow authors Eric, Jamie, and Tom, it has been a pleasant and challenging experience. Thanks for not giving up at the outset! Even though the team changed as we went along, the very real need for this book to be made kept it going.

To my fellow coworkers at Walter P. Moore, thank you. I am the first to admit that what I know is because of the people I work with and the projects I work on. We share our skills and knowledge freely, and it makes us collectively so much better for it.

To Autodesk and the Revit Structure team, thank you. Your continued efforts to provide a solution to the structural field gave us something to write about! Not to mention, it is fun modeling buildings! Thank you, Nicolas and Wai, for your support and enlightenment.

To the publisher, Willem, thanks for your encouragement and belief in our concept and desire to write this book. Without our advocate none of this would be possible. Special thanks to Tom for the guidance as I ramped up and began the work and for the patience as I struggled to finish!

—David J. Harrington

About the Authors

Thomas S. Weir

Tom is associate principal and director of BIM & CAD Operations at Brandow & Johnston, Inc., a consulting structural and civil engineering firm in Los Angeles, California. He has almost 30 years of structural design experience on numerous architectural-engineering building projects both large and small.

An early adopter of Revit Structure modeling software and a longtime modeling enthusiast, Tom continues to be in the vanguard of those seeking to help transform the AEC industry as it transitions into the new BIM design era. He is cochairman and founder of the Los Angeles Revit Users Group, one of the most dynamic user groups in the United States. Tom also helped start the AUGI Revit Structural forum.

His first book is used widely for training, *Autodesk Official Training Courseware (AOTC) Revit Structure 4 Essentials*. Tom is a frequent lecturer on Revit Structure and building information modeling (BIM) and has taught classes at Autodesk University for the last several years.

Tom grew up north of Boston, Massachusetts. After high school and some college, he did a tour in the U.S. Army, leaving as a sergeant in the military police corps. He then studied at UMASS Amherst, where he received his BA in philosophy with minors in english and education. With few jobs available for philosophers, he went to engineering school at Northeastern University in Boston, got married, started a family, and eventually moved cross-country to California, where he began his 27-year tenure at Brandow & Johnston.

In his spare time Tom likes to camp with his family. Music and Astronomy are his main hobbies. He likes to play all sorts of American roots music on his Martin D-18 guitar.

Jamie D. Richardson

Jamie is an associate and CAD/BIM manager at Ericksen, Roed and Associates, a structural engineering firm based in Saint Paul, Minnesota. He has collaborated with several of the architectural firms in the Twin Cities on multiple Revit Structure projects. Jamie joined Ericksen, Roed and Associates in 1996 as a structural designer and, over time, completely modernized its AutoCAD customization.

Throughout his 14 years of using Autodesk products, Jamie has been instrumental in the rollout of several versions of AutoCAD as well as the implementation of Revit Structure. His responsibilities include oversight of all Revit Structure production. Jamie has been a beta tester since RS2, an avid speaker on Revit Structure at Autodesk University, and a contributor to the Revit Structure forums on AUGI.

His local Revit Structure involvements include being a member of the Minnesota Revit User Group (MNRUG), participating in other speaking engagements on building information modeling collaboration efforts, and mentoring students at local technical colleges.

Outside of work, Jamie enjoys spending time with his family at their cabin in northern Wisconsin. There he likes to fish, play on the water, and relax by late-night campfires.

Eric Wing

Eric Wing is a CADD/BIM support specialist for C&S Companies in Syracuse, New York. Eric has been in the architectural engineering industry since he graduated from Delhi University in 1991. Eric is also the director of the AUGI Training Program (ATP) and is a monthly columnist for various publications. He is also a popular speaker at Autodesk University and many other national events.

David J. Harrington

David is a senior associate with Walter P. Moore and Associates, one of the premier consulting structural engineering firms in the United States. He has over 21 years of structural drafting and design experience on projects ranging in size from a convenience store to an NFL stadium and convention center covering millions of square feet.

He has been working with Autodesk products since 1987, starting with AutoCAD and later delving into 3D Studio Max and Architectural Desktop, and with Tekla Xsteel (Structure). David has also been customizing the AutoCAD working environment with AutoLISP and other interfaces to aid in controlling and managing standards for Walter P. Moore. He began using Revit Structure at version 1 and conducts in-house training and customization for this application.

David has written or coauthored for many years. He created the *PaperSpace* newsletter produced first by the North America Autodesk User Group (NAAUG) and then by Autodesk User Group International (AUGI). He then began assisting in the editing arena and is the current technical editor for *AUGIWorld* magazine. Books he has worked on are *Inside AutoCAD R14*, *Inside AutoCAD 2000*, *Inside AutoCAD 2000i*, *Inside AutoCAD 2002*, and *Inside AutoCAD 2005*.

Back in 1994, David was elected to the board of directors of NAAUG, where he served as the local user group representative. Later he was elected as the AEC industry chair; then within AUGI he was elected to the position of president and served in 1998–1999. Other major contributions during these times are the AUGI Guild, an email-based support system for Autodesk users, and the formalization of the Wish List into a web-hosted system for real-time voting.

He has also been an instructor at Autodesk's annual training event, Autodesk University, teaching classes on Revit Structure adoption and other Structure-related subjects.

David has lived nearly all of his life in the Tampa area of Florida. In his spare time David enjoys wine and an occasional cigar. His hobbies are limited to relaxing and computer gaming.

Contents at a Glance

Contents

Introduction

Thank you very much for buying this book. The four authors have been working tirelessly over many months to create something that will appeal to advanced and novice users alike. We are proud to say that this is the first major book published on this subject.

Though the book is weighted toward the structural discipline, we think this book will also appeal to architectural and MEP designers and drafters as well, since much of the same functionality exists in all three Revit versions. We authors have all been early Revit adopters and have done scores of projects in real production over the past several years. We have taught Revit, written articles and blogs, and managed user groups, so we have a good idea of what you might want to learn. Coming from different regions of the country also helps give each of us a different perspective on the subject.

For the most part we focus on architectural-engineering building projects rather than civil engineering projects. Most important, we have deliberately tried to filter the material through our experience both as project managers and as teachers. In that way we have attempted to distill the large amount of subject matter down to a manageable set of information so that you can focus on what will be most useful for you as you begin to tackle your own projects. So this book will not cover every single option of every single command and function that you find in Revit Structure, but it will try to point out the most useful items that the filter of experience has taught us you will need, and those are represented in real-world project scenarios.

For more on any of those items not covered you should refer to the documentation in the Revit Structure Help menu and tutorials. The documentation has gotten better and better in the last few years, and you need to refer to it often to keep learning in the most productive and efficient way.

If you are new to Revit Structure, a little bit of history is in order so that you are able to appreciate how far this program has progressed since it was purchased by Autodesk in 2003. At that time all of the tools you will learn about in this book were hard to use, if they existed at all. The original Revit developers focused on the architectural industry, and so the structural portions of the program were never fully developed. When Autodesk acquired the Revit product and company, all that changed. Significant resources were allocated to get Revit development up to speed, and soon the Structural module was released. In a few short years Revit has become the 900-pound gorilla in the BIM world, accounting for nearly 70 percent of the BIM AEC software market. With each new version of the program, and most notably when Revit Structure was developed as a separate application, it has become more complete in its capabilities. The developers at Revit Structure have constantly expanded its functionality. They definitely have been listening to the community of users and have worked very hard to provide new tools to us end users on a timely basis.

They get it! Autodesk invested heavily when BIM was in its nascent period and helped create the massive transformation that we see today in the AEC industry. Back in the 1990s when 2D computer-aided drafting started changing the industry, there were many drafters who said they did not need to learn it because hand drafting would always be available. By the late 1990s their jobs were all but gone. Today we are in a similar situation. Some say they do not need to learn about Revit Structure or BIM because there will always be 2D computer-aided drafting available. Those shortsighted people will soon suffer a fate similar to what the hand drafters suffered.

The chapters in this book are broken down into five parts that guide you from the moment you click on the program icon to first open the program all the way to advanced concepts such as family creation and design options. After covering the basics of the modeling environment, you move on to learn how to create a three-dimensional model. After that you learn to document and share your model with others. Finally you take an in-depth look at advanced modeling topics.

There is also a wonderful color gallery in the middle of the book that shows off some of the many and varied design projects done by the authors. One of the most unique parts of the book is an appendix that goes into even greater detail in describing some of the gallery projects.

So if you are a Revit Structure novice, do not wait; dive in and join this exciting march into the future of building design. If you are a Revit Structure expert, you still will find many interesting concepts and procedures here that you might not have heard about before. Revit Structure rocks!

What You Will Learn

In this book you will learn the basics as well as more advanced techniques used to create a BIM model for a structural engineering project using Autodesk Revit Structure software. You will learn how to prepare construction documents after you have developed your model and how to collaborate with others by linking models or exporting to AutoCAD. You will learn how to detail and schedule the elements in your virtual building. In the last portion of the book, advanced subjects such as standards, rendering, and creating structural families will add a new dimension to your knowledge and abilities.

What You Need

Some knowledge of Revit Structure will be very helpful. Some knowledge of how buildings are designed and constructed will be helpful as well.

The *Mastering* Series

The *Mastering* series from Sybex provides outstanding instruction for readers with intermediate and advanced skills, in the form of top-notch training and development for those already working in their field and clear, serious education for those aspiring to become pros. Every *Mastering* book features:

- The Sybex "by professionals for professionals" commitment. *Mastering* authors are themselves practitioners, with plenty of credentials in their areas of specialty.
- A practical perspective for a reader who already knows the basics—someone who needs solutions, not a primer.
- Real-world scenarios, ranging from case studies to interviews, that show how the tool, technique, or knowledge presented is applied in actual practice.
- Skill-based instruction, with chapters organized around real tasks rather than abstract concepts or subjects.
- Self-review test "Master It" problems and questions, so you can be certain you're equipped to do the job right.

Who Should Buy This Book

All those structural, architectural, and MEP people interested in learning about Autodesk Revit Structure and building information modeling should read this book. The text is geared toward all levels while trying especially hard to cover subjects beyond the basics that will appeal to mid and advanced users. Many real-world project scenarios are discussed as well as actual projects and how they were modeled.

This is a book by experienced, power Revit Structure users and not salespeople. If you are looking for a book that tells both good and bad about this subject without the sugar coating that sale types often give you, then this book is for you. We pull no punches and look under every rock in our effort to expose the underlying reality of the situation.

What's Inside

Here is a glance at what is in each chapter of *Mastering Revit Structure 2009*:

Part 1: Basics of the Modeling Environment

Chapter 1: Inside Revit Structure

The basics are described in this first chapter, such as the layout of the different menus and commands. You learn about the types of elements in the modeling environment and how to create and manage project views. Another important area that is examined is how to control the graphical display of your modeled elements in a project.

Chapter 2: Setting the Project Environment

Revit Structure, like any other program, has settings that help control the environment that you will be working in. Chapter 2 shows you how to develop different project environments through the use of templates. We discuss the various settings that are available to you as well as how the Project Browser can be organized so it can be managed. Then we discuss how content and settings can be transferred from one project to another.

Chapter 3: Starting to Model Your Project

This chapter explores the basics of getting a structural model started. Importing and linking CAD data and linking Revit Architecture models are explained and demonstrated. You then learn to establish your grids and levels and use the copy/monitor feature in order to coordinate with the architect.

Part 2: Developing Your Structural Model

Chapter 4: Structural Columns

Before we start talking about placing structural columns in Chapter 4, we talk about the basic templates used to create them as well as the various parameter settings that are available to control their behavior. We explain things you can do after they are placed that will allow the columns to adapt to other changes in the model. Modeling sloped columns is not yet part of the program in Revit Structure, but we show you ways to model them with other tools. And last we give you an in-depth explanation of everything you wanted to know about the Graphical Column Schedule for scheduling columns.

Chapter 5: Floor Slabs and Roof Decks

In this chapter you learn to create many different types and shapes of slabs, floors, and roofs to add to your building model. Composite deck creation is covered as well. So-called flat roofs that really have minor sloping from ridges to drains are especially difficult to model. You will see how Revit Structure handles theses cases by using the sub-element tools.

Chapter 6: Walls

Walls are system families in Revit Structure and are given plenty of attention in this chapter since they are a fundamental element of most projects. How to create walls, how to place them into your model, and how to edit them as the design process evolves are all covered.

Chapter 7: Structural Framing

After creating the floor slabs and roof decks in Chapter 5, you now learn how to add support framing beneath them. Basic floor framing tools are explored. Creating sloping roof framing is an especially interesting part of this chapter. You also explore framing families and their properties. We discuss how to add moment and braced frames to a project to round out this challenging subject.

Chapter 8: Foundations

As your project develops, you will have to add foundation elements to support your structures, and they come in many forms (no pun intended). In this chapter you will learn to create strip and step footings. Foundation slabs and grade beams are discussed as well as how to construct a basic elevator pit in your project.

Part 3: Documenting Your Structural Model

Chapter 9: Model Documentation

Now that you have the model up and running, you need to know how to add the notations to the various views you have created.

First you will study the datum elements and how they are added and controlled in your detail and section views. Next is a discussion of annotation elements such as text, tags, and symbols. You will examine how to add detailing elements such as detailing lines and filled regions to your views. Finally you will learn how to create a typical details library.

Chapter 10: Modeling Rebar

Most of the construction in the world is done in concrete, and so Revit Structure must have a very robust system to incorporate it into your virtual model and your construction documents. You will learn how to configure the rebar settings and then how to draft 2D rebar. Then you will examine how 3D rebar is used and how new shapes can be made.

Chapter 11: Schedules and Quantities

To achieve a BIM solution, you will find that your ability to extract information from your model is crucial. Here you learn to do just that by creating schedules and material takeoffs. You will also see how to export schedules to Microsoft Excel. Legends are another type of schedule that is examined.

Chapter 12: Sheets

In Chapter 12 we talk about creating sheets with title blocks to help organize your views and as a way to document your model. Title blocks usually include revision schedules, so we show you how to create and incorporate them into your sheets so Revit Structure can manage revisions made to the model. We also discuss creating sheet indexes and how the properties of views may change as they are placed on sheets.

Part 4: Sharing Your Structural Model

Chapter 13: Worksharing

Regardless of a project's size or the number of team members working on it, you still might want to switch from using a single-user file to a multiuser environment. In Chapter 13 we discuss when it might be right for you to enable worksharing and walk you through the steps to do so. We also talk about the typical workflow and the tools used to help you work and communicate with other team members.

Chapter 14: Visualization

After spending a considerable amount of time modeling your projects, you will want to gain admiration for the work. To do this you will need to render them! In this chapter you will learn how to evaluate what and when to model. From there you will explore the process of learning and operating the rendering engine Mental Ray in Revit Structure. Lastly you will uncover other options for extending your model use after you export it.

Chapter 15: Revit Structure Analysis

This chapter provides a close look at using the analytical model and related commands. You will learn how to configure the analytical structural settings and create loads for your project. You will examine how to place analytical load patterns onto your model. Finally you will learn how to import and export your virtual model from Revit Structure to structural analysis software.

Part 5: Advanced Topics

Chapter 16: Project Phases and Design Options

This chapter takes you to a new level of mastery in your ability to manipulate the model by adding phases and design options. You will examine a real-world example of a historic residence where existing and new phases had to be presented to the historical commission. Added to that were several different design possibilities. All this was done in one Revit Structure file.

Chapter 17: Standards: Increasing Revit Productivity

You have just upgraded to Revit Structure and started to create models. But have you thought about your old AutoCAD standards? Using the information in Chapter 17, you will learn what can and cannot be done easily in Revit Structure with regard to standards. Afterward you will delve into enhancing your work through Revit Structure customization. Then you will actually implement your new model standards and apply view overrides effectively.

Chapter 18: Family Creation: Beyond the Built-In Libraries

Family creation surely will take you much deeper into understanding the power of Revit Structure. You will learn to create a footing step family and in-place families. Groups are another powerful tool that you will find indispensable in your work. These subjects will give you a whole new perspective on modeling.

Chapter 19: Advanced Structural Families

This chapter takes you to the next level of understanding of the family-creation process. You will study the development of several important structural families and how the families are used. Instead of building your elevator pit from scratch each time, you learn here how to develop an elevator pit family that can be inserted directly into your project.

Appendices

Appendix A: Master It Solutions

Every chapter ends with a collection of short exercises designed to reinforce the essential skills taught in the chapter. This appendix provides the solutions to those exercises.

Appendix B: Modeling Project Types

This appendix expands and explains the development of some of the complex projects that are presented in the color gallery section. You will see how different project types are created using Revit Structure and what difficulties were surmounted.

How to Contact the Authors

Sybex strives to keep you supplied with the latest tools and information you need for your work. Please check the website at `www.sybex.com/go/masteringrevitstructure2009`, where we'll post additional content and updates that supplement this book if the need arises. Enter **Mastering Revit Structure 2009** in the Search box (or type the book's ISBN—9780470384404), and click Go to get to the book's update page.

Mastering

Revit® Structure 2009

Part 1

Basics of the Modelling Environment

Chapter 1

Inside Revit Structure

The Revit Structure interface is designed to be an easy-to-use, organized presentation of commands and drawing areas. The basic interface is highly configurable and can be adapted to fit most working needs that may arise as you interact with the computer in order to create a model. Multiple model views can be open at once so that you can see plan, section, elevation, and model views of an element side by side in the drawing area at one time.

This chapter will explore the interface and the arrangement of menus, toolbars, tabs, and other tools available to you in the Revit Structure workspace. The tools found here will be used to model and document your virtual structure, and also help you to achieve a building information modeling (BIM) solution for your project.

Underlying the graphical interface is a robust database that coordinates the graphical information through the use of a parametric change engine that controls the display of all elements in your project. In this way, a change in an element that takes place in a plan or a section view is immediately propagated to all views in the project, which in turn save lots of time for you and allows you to focus on the design of the structure rather than the busywork of having to edit many views in order to correct one element, as occurs in most 2D drafting programs.

To succeed in this venture, you will need to add many tools to your tool chest. Imagine a construction worker who goes to work with only a hammer and a screwdriver. In all likelihood the worker will not be able to get much accomplished. So to get started, you need to add the necessary tools to your own tool chest so that you can work effectively in the virtual model-building environment.

Once you have learned the basics of the various display and modeling tools available to you in Revit Structure, you will be ready to move on to the actual modeling process in the subsequent chapters of this book.

In this chapter you will learn to:

- Use the graphical user interface
- Understand the types of elements in the modeling environment
- Create and manage project views
- Control the graphical display of elements in a project

Using the Graphical User Interface (GUI)

When you double-click the Revit Structure 2009 icon on your desktop, you open the preview dialog box. This dialog box shows projects and families on which you have recently been working. Click one of the existing file icons to open them, or click New to begin a new project (see Figure 1.1).

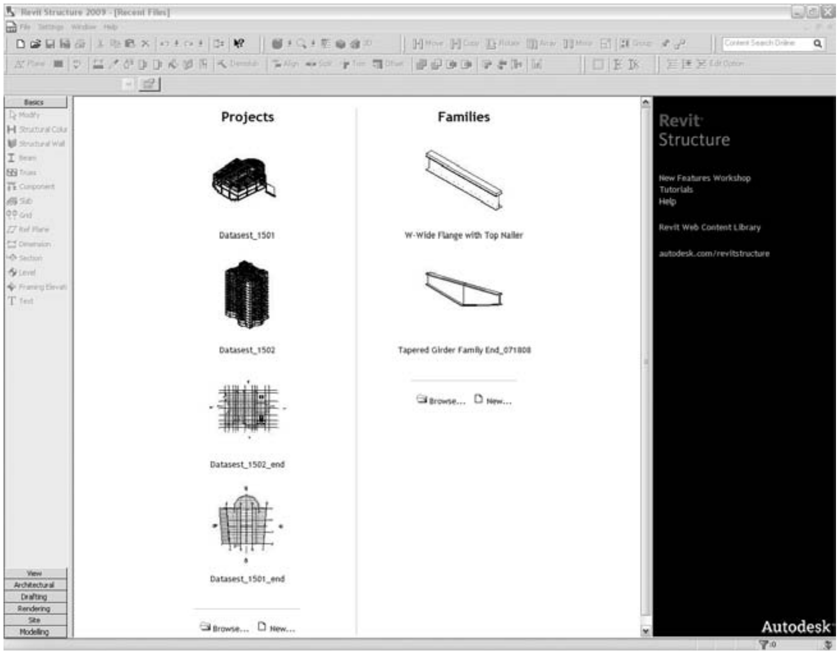


FIGURE 1.1 Opening dialog box with graphical file display

If you click New, the New Project dialog box will appear. In that dialog box you either choose to create a new project template or use a specific project template. Click OK, and then Revit Structure puts you into the main graphical user interface (GUI).

The Revit Structure GUI (see Figure 1.2) is task oriented, with a compact footprint and semi-rigid structure. The task-oriented layout and grouping of like commands makes project management and modeling within Revit Structure logical and efficient. The compact footprint of the GUI is due in large part to the lack of excessive toolbars and dialog boxes, which tend to clutter other platforms. This allows a large area in which to work and view the model. The location and size of most GUI components are reset to their defaults when you restart Revit Structure, making the interface somewhat rigid with regard to user customization.

The GUI (see Figure 1.2) contains a menu bar, toolbars, Options bar, Type Selector, Design bar, Project Browser, drawing area, View Control bar, and a status bar, all described in the following sections.

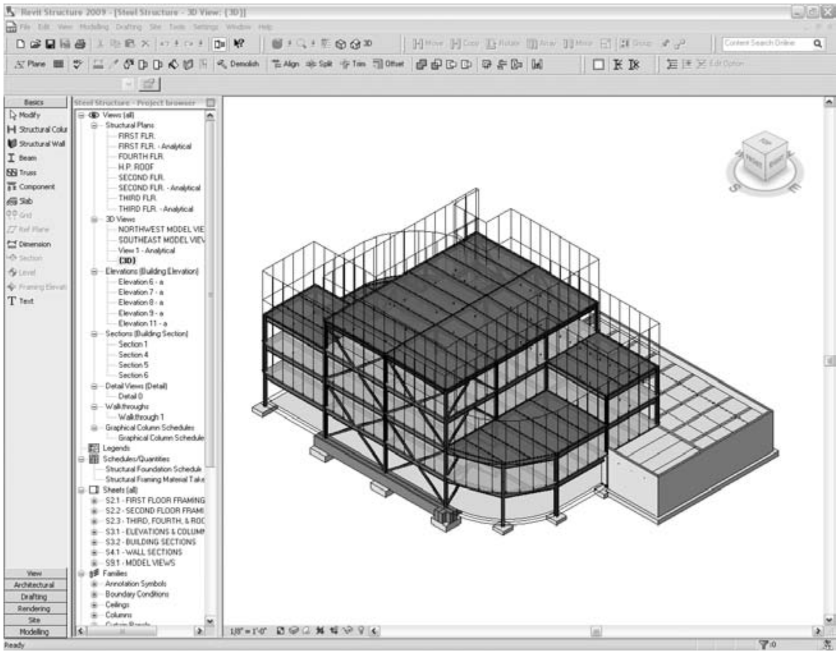


FIGURE 1.2
Default GUI with its intuitive display of commands

Menu Bar

The standard Microsoft Windows-based menu bar (see Figure 1.3) is located at the top of the GUI and provides direct access to all of the commands and settings available in Revit Structure. Commands with defined shortcuts will have the shortcut key shown to the right of the command name in the menu.

FIGURE 1.3
The menu bar is an easy-to-use pull-down arrangement of most commands.

Toolbars

The default Revit Structure toolbar layout (see Figure 1.4) is located directly below the menu bar and houses frequently used commands for file management, view navigation, and element modification. Toolbar and text label display is controlled from the menu bar: Window ➤ Toolbars or the right-click context menu. Display can also be handled by editing the `Revit.ini` file, which will be discussed in a later chapter. Hiding all of the toolbars will completely remove the toolbar area from the interface. To show them again, you will need to go through the menu bar or edit the `Revit.ini` file.

Figure 1.4 The toolbars contains many frequently used commands.

You can rearrange toolbars by left-click-dragging the divider located at the beginning of each toolbar. While toolbar and text label display is stored, the arrangement of the toolbars is not. All toolbars set to display will be reset to their default positions when you restart Revit.

Options Bar

The Options bar (see Figure 1.5) is located directly under the toolbars and displays options specific to the active command or selected element(s). These options can be huge time-savers by negating the need to dig through a dialog box or launch another command. Even experienced users can increase productivity by keeping a watchful eye on the constantly changing Options bar and making use of the controls provided. The location of the Options bar above the display area makes it easier for the eyes to notice changes. You do not have to keep glancing up and down as much as you do in other applications.

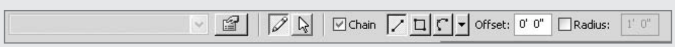
Figure 1.5 Keep an eye on the Options bar as its commands change.

Wish Granted Courtesy of the Options Bar

Both beginner and experienced users of Revit Structure alike have uttered, "I wish Revit Structure could..." or "I wish Revit Structure had..." among other colorful expressions. These wishes can often be granted by simply looking to the Options bar. The following graphics illustrate the many looks of the Options bar during the execution of a single modeling command.

When you first invoke the Slab command, the default placement method is Pick Walls, and the Options bar shows various default options, as shown in the following graphic:

If you prefer to place the slab by sketching lines, you select the Lines command and the Options bar will change, as shown here:



You finish the slab and select it in plan, and the Options bar now shows editing controls:

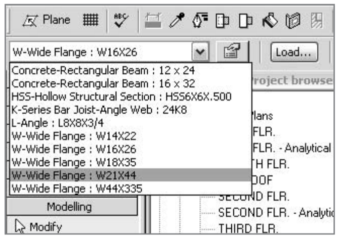
As you can see, the Options bar had three radically different configurations, each loaded with context-specific controls that in many cases cannot be found anywhere else. In most cases, after you initiate a command you will want to look to the Options bar.

Type Selector

The Type Selector (see Figure 1.6) is located on the left end of the Options bar and lists all of the types available for a given element that are loaded into your project file. Generally you should only load in the types you need to use in order to keep the computer performance from slowing. The Load button is located conveniently close to the Type Selector so you can load in more types as they are needed. Why load every type of wide flange steel beam into your project if you only need a W12x26 and a W18x40?

The Type Selector is used to select a specific element type for placement. If you highlight an element that has been placed in your project, you can then use the Type Selector to identify it or change its type—for instance, to change a W12x26 to a W18x40. Within your project views any existing beam tag for that member size will automatically change its corresponding value to the new type.

FIGURE 1.6
The Type Selector



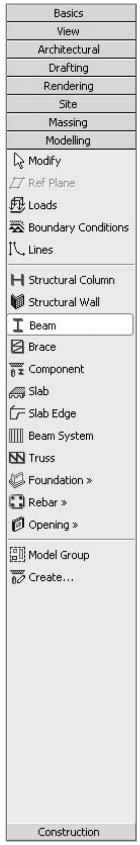
Design Bar

The Design bar (see Figure 1.7) is permanently docked along the left side of the GUI and contains commands that are used for modeling, annotating, and viewing. These commands are organized into nine task-specific tabs: Basics, View, Architectural, Drafting, Rendering, Site, Massing, Modeling, and Construction. The View, Drafting, and Modeling tabs contain the majority of the commands required to create and document a structural model. The Basics tab combines the most commonly used commands from these three tabs and is an ideal tab to have active for most modeling and annotation tasks.

In addition to the nine tabs, two context-specific tabs (Family and Sketch) will appear and remain active for the duration of certain commands. The Family tab is used to create and edit in-place and external families. The Sketch tab is used to create and modify a sketch and will be displayed on the Design bar when using the Slab or other sketch-based elements.

Tab display is similar to toolbar display in that it is controlled from the menu bar: by choosing Window ➢ Design Bars or by right-clicking on the Design bar and selecting a tab name from the context menu. Tab display can also be handled by editing the `Revit.ini` file, which will be discussed later in Chapter 17. Adjust the width of the Design bar by left-click-dragging the right border. Tab display settings are remembered, but the width of the Design bar will be reset to its default when you restart Revit Structure.

FIGURE 1.7
The Design bar houses tabs that organize your commands.

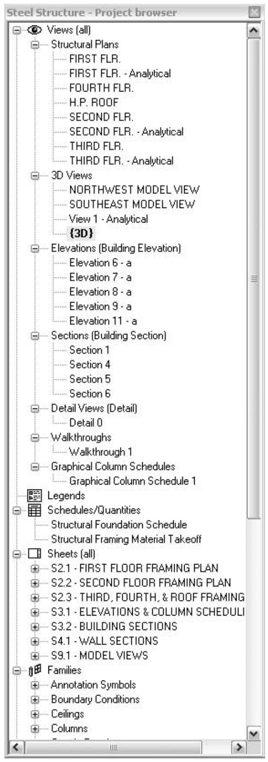


Project Browser

The Project Browser displays all of the views, families, groups, and Revit Structure links in a Windows Explorer–style format, and is located to the right of the Design bar (see Figure 1.8). The views in the Project Browser can be sorted, grouped, and filtered in a variety of ways depending on how you want to organize your project. The organization will be discussed more fully in the next chapter.

Unlike the Design bar, the Project Browser can be docked on the left (its default location), top, right, or bottom of the drawing area. You can also undock and resize it by left-click-dragging any of its borders to act as a floating dialog box. The Project Browser will return to its default location and size once you restart Revit.

Figure 1.8
Project Browser

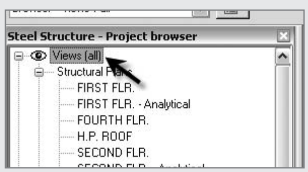


Exercise: Create a New Browser View Type: On Sheets

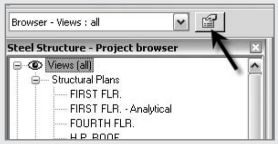
This exercise will step you through the process of creating a new browser view type. The new browser view type will be called On Sheets, and as the name suggests you will only be able to see views in the project that are placed on a sheet. While working on a project, numerous working, construction, and coordination views are typically created and can clutter the Project Browser, so this view type can be quite useful.

This browser view type will allow you to easily view and work on only the views that are part of your construction documents.

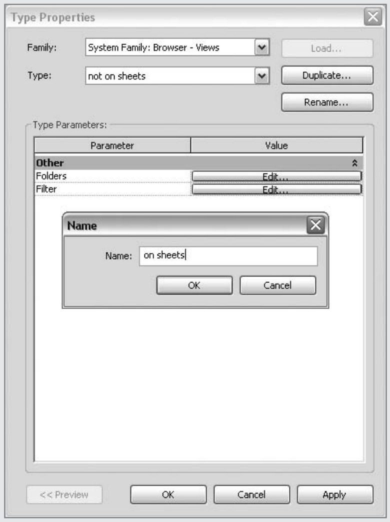
1. Start a new project.
2. Select the Views heading in the Project Browser, as shown in this graphic:



3. Click the Element Properties button to the right of the Type Selector, as shown here:

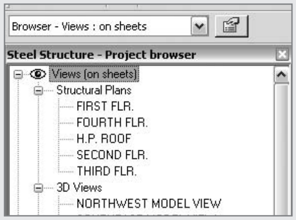


4. From the Type drop-down menu in the Type Properties dialog box, select the Not On Sheets type.
5. Click Duplicate, then type **on sheets,** as shown below, for the name and click OK to close the Name dialog box:



6. Click the Edit button next to the Filter parameter.
7. In the Browser Organization Properties dialog box, you will see that the Filter By drop-down menu is already set to Sheet Name, followed by a drop-down menu set to Equal To.
8. Change the Equal To drop-down menu to Not Equal To, as shown here, and click OK to close the dialog box:

9. Finish by clicking OK to close the Type Properties dialog box. Note that your browser is now sorted by your new browser view type as illustrated below:



You can change your browser view type at any time by selecting the Views heading in the Project Browser and selecting a different type from the Type Selector. The Revit default browser view type is All.

Drawing Area

The drawing area (see Figure 1.9) is where all open views are displayed. Multiple views of several projects can be open at one time. Project templates and families can also be open in a single session of Revit and displayed in the drawing area. You can arrange all of these open views in the drawing area by using the Window menu bar in typical Windows fashion. All the view controls you would expect are here, including Tile, Cascade, and Arrange, which perform as you would expect.

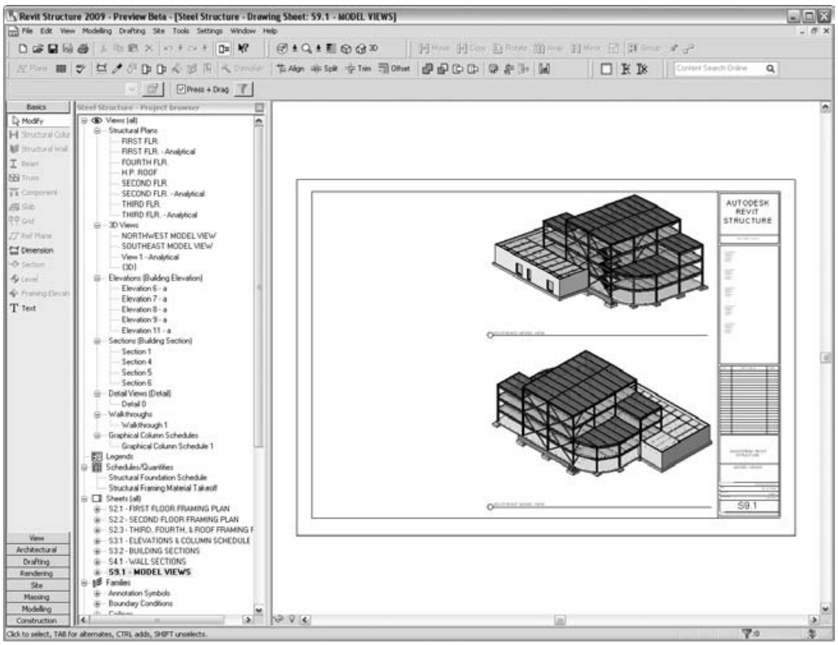


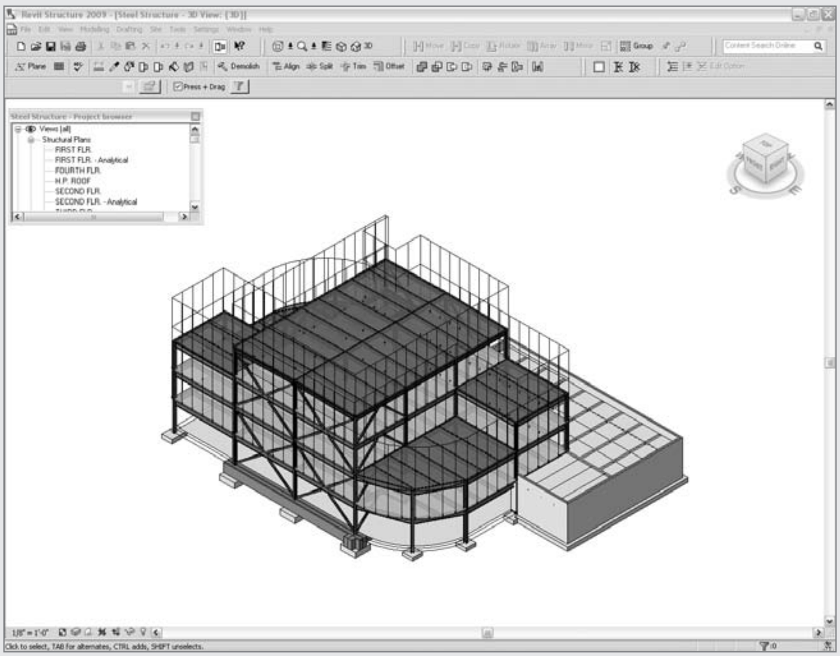
FIGURE 1.9
The drawing area contains all your open views, which can be arranged in many ways.

Having many views open at once, though, can lead to a slower program response. The Close Hidden Windows command is quite important in managing the performance of Revit, especially when you are working on large or complex projects. With one view maximized in the drawing area, this command will close all but the last active view of each project, project template, and family that is currently open.

Most views open maximized in Revit regardless of how they are triggered, so it is easy to quickly have numerous views of a project open at any given time without noticing it. Since Revit updates the display of all open views in real time, you can imagine how this can rapidly become a performance issue. Frequent use of this command is highly recommended and is a good candidate for addition to your shortcut list.

Presentation GUI

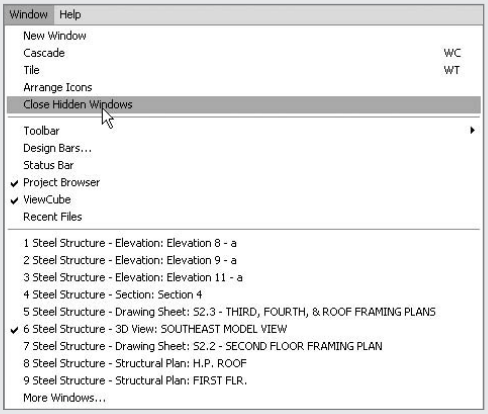
While most will find the default interface more than adequate for daily use, a few simple modifications can drastically change the look of the GUI to better serve other purposes or user preferences. The following graphic shows the interface with the Design bar and toolbars hidden and a floating Project Browser. This clean look is great for viewing and working with the model while in meetings with the design team and/or clients.



Real World Scenario

Nice Postage Stamp Collection

I got a new PC recently and was working on a rather complex healthcare project. Impressed with the improvement in performance compared to my previous PC, I had not been paying attention to how many views I actually had open. The PC slowed considerably (a several-second pause instead of the expected near instantaneous response) while I was attempting a rather routine procedure. It was not until I tiled my open views a few commands later that I realized what the problem was: 25 open windows! They looked like a bunch of postage stamps. As I began to use the Close Hidden Windows command, as shown in the following graphic, my PC's performance returned to normal.



View Control Bar

The View Control bar (see Figure 1.10) is located on the bottom left of most views. The controls found on the View Control bar are for commonly used view and element display properties. Each button expands when you select it and displays several context-specific settings. There are eigh main tabs on the bar:

- Scale
- Detail Level
- Model Graphics Style
- Shadows
- Crop Entire View On or Off
- Crop a View Region
- Temporary Hide/Isolate
- Reveal Hidden Elements

Figure 1.10
The View Control bar makes it easy to control the view display.

Working views can change from minute to minute, for instance as you change between coarse and medium detail modes, so it is very convenient to have the View Control bar nearby for easy access to the controls.

Status Bar

The status bar is located at the very bottom of the GUI. The text on the left of the status bar will do the following:

- Display the name of a highlighted element.
- Display prompts and/or additional information regarding the active command.
- List the available shortcuts that are available for a given sequence of characters, which can be navigated using the arrow keys.

On the right end of the status bar, you can view the status of your keyboard locks (Caps, Num, and Scroll). To the right of this is an element selection counter, which will display the number of elements currently selected in the drawing.

The Communication Center is located at the right end of the status bar and can be configured to check for software updates, product support announcements, and other related news at intervals defined in its settings. The Communication Center can be opened by double-clicking or by right-clicking the icon. In the Communicaton Center dialog box that is then displayed you configure which information channels you want to receive information about, such Articles and Tips, and at what interval you wish to update your local workstation with the information.

Shortcuts

Shortcuts are one of the most powerful ways to increase your overall speed and productivity when using Revit Structure. Shortcuts enable you to launch commands directly from the drawing area without having to move your cursor. That significantly reduces mouse travel, saving time and your wrist. Many shortcuts are already defined, and you can create others for nearly all commands found within the menu bar.

Right-Click, or Context, Menu

The right-click, or context, menu provides quick access to useful commands directly related to the element or object being highlighted. As previously discussed, right-clicking on GUI objects (toolbars and the Design bar) displays a menu that controls how they are displayed:

- Right-clicking in the empty space of a view displays a menu (see Figure 1.11) with view navigation commands and access to the properties of the view.
- Right-clicking different elements in the model will also display various menus (see Figures 1.12 and 1.13) with access to element- and view-specific commands and properties.

FIGURE 1.11
View context menu

FIGURE 1.12
Framing element context menu

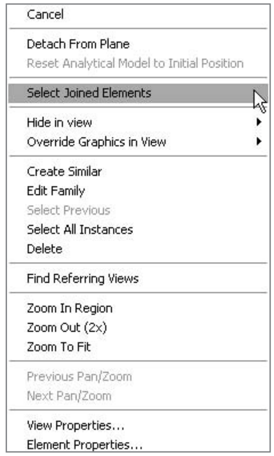
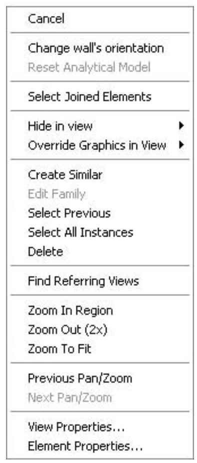


FIGURE 1.13
Wall element context menu



Now that you have had a good look at the basic Revit Structure interface and command layout, let us explore the various elements that these commands create and manage in the creation of your building project.

Elements

One of the important things to understand about Revit Structure in its approach to modeling is that it is object oriented, rather than line based as in traditional 2D drafting. Instead of drawing a series of lines on a flat sheet to represent a column, you go to a virtual library, load a column element, and then place it in your virtual working space. That column displays in every view. In addition to modeling elements, other element types are available to help you document your design, as we will discuss in this current section.

Three types of elements are used to model and document a project in Revit Structure: model elements, datum elements, and view-specific elements. These elements are organized in order to allow you to easily control their on-screen and printed appearance and display. Let's take a look at these elements and how they function.

Model Elements

Beams, columns, walls, and other real-world building objects are represented in Revit Structure by model elements. These are the primary elements used to create the model and are typically placed as they would be constructed. This approach allows accurate quantities and views to be derived from the model.

Model elements can be altered in any view in your project in which they appear. Once changed, every related view is automatically updated by the underlying database. This is called *bidirectional associativity*, and is one of the most important aspects of Revit Structure.

One of the promises of BIM technology like Revit Structure for project design is that much of the busywork of creating construction documents will be reduced so that you can concentrate on the design of the project rather than wrestling with the design software. The use of model elements is a good case in point.

Two distinct types of model elements exist in Revit Structure:

Host Host elements are generally system families (which will be discussed later in this chapter) that represent real-world construction elements such as walls, slabs, roofs, and stairs. These elements, as their name implies, often host other elements such as openings in a wall or reinforcement in a slab.

Component Component elements are used to represent all other real-world construction elements, including beams, trusses, columns, and reinforcing bar. These elements are typically external families that are loaded from the Revit Structure libraries into the project as needed, similar to their real-world counterparts being trucked or shipped to the site for assembly.

Most of your modeling will use these elements, so it is important for you to understand their basic properties.

Datum Elements

Grids, levels, and reference planes are datum elements. These elements provide the framework in which the building elements are placed and flexed. As you add your model elements to the

project they will become fixed to the datum elements. These basic modeling constraints then become anchors for objects so that if you need to change a bay width or the story-to-story height of a level those elements will also move correspondingly. For example: beam elements placed in a third floor plan view are associated with that datum. Changing the elevation of the level will take the beams along for the ride as they are defined as belonging to that level. This makes floor-to-floor clear height adjustments quick and accurate.

Datum elements are an essential part of a constraint-based modeling system and are fundamental to how you will assemble and edit the design.

View-Specific Elements

View-specific elements are used to annotate and detail specific views of the model for the creation of your construction documents:

Annotation Annotation elements include text notes, tags, keynotes, dimensions, spot elevations, spot coordinates, and symbols. These elements play a critical role in translating the model into construction documents. Unlike simple annotations found in other platforms, the majority of the annotation elements in Revit Structure have a great deal of intelligence. Tags, for example, are annotations that display specific parameter values contained in the model elements. Change the size of a beam in the model, and all tags that you have already placed will be updated automatically. Adding text is also a view-specific element (see Figures 1.14 and 1.15).

Detail Detail elements pick up where the model elements leave off. Some items are not worth the time, effort, or performance overhead to model and can easily be handled with the addition of simple 2D line work or by adding 2D detail components, such as to a section cut through the model. These elements are used to complete in 2D the areas that are not modeled but whose display aids in showing design intent. For instance you might add 2D earth hatching around a foundation footing.

How Much Should I Model?

How much should you model and how much should you just add in 2D?

You need to ask yourself this question quite often as you proceed with modeling your project. You might model the columns but not model the base plate and bolts in your project if you work for a design engineering firm. In that case, modeling a few typical cases of various connection types will be sufficient.

On the other hand, if you work for a detailing or construction management firm, you might have to model every piece in the structure. The scope and extent of your model building depends on the documents that will be derived from it, as well as the BIM solution you are trying to achieve.

The bottom line to knowing how much to model in your project is that you must maintain the essential integrity of the model by creating and maintaining the necessary elements to suit your purposes.

FIGURE 1.14
The gray line work consists of model elements.

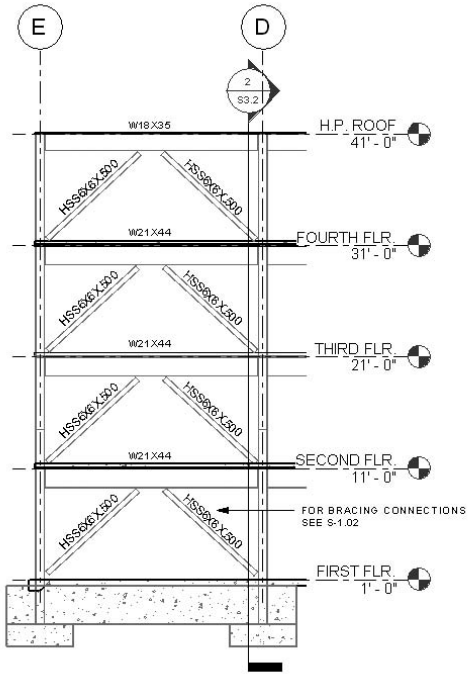
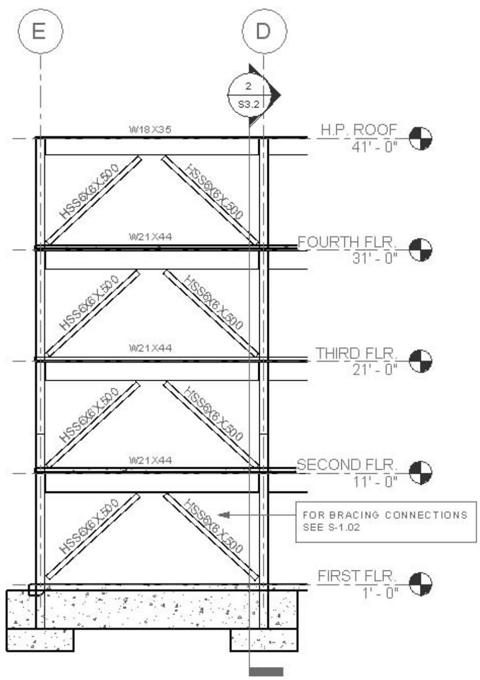


FIGURE 1.15
The gray line work consists of detail and datum components.



As the name implies, View-Specific elements only exist in the view in which they are placed, with dependent views the exception to the rule. Dependent views are child views to a single parent view and share all view-specific elements with the parent and its other children. An example of a dependent view is a large framing plan that needs to be divided into several sections so it will fit onto your title sheet.

Model elements, on the other hand, appear in every view whose extent they intersect. That saves a lot of time and coordination effort when you are making significant changes in the design. And remember you can work on the model elements in any view in which they appear.

Element Organization

All of the elements used throughout Revit are logically organized into a hierarchy of categories, families, types, and instances:

- The categories represent the different parts of the building, such as structural foundations, columns, and beams.
- Within each category are different families of objects. The structural column category has steel column families and concrete column families.
- Within each family are different types of the same object,.The steel wide flange column family has many sizes, such as a W12x26 and a W24x55. These are different types of one family.
- Each type can be placed many times in your project, and with various settings, which we call instances of a type. One instance may be a one-story column; another instance may be a four-story column.

Next we will examine these different parts of the element organization and how they work together in your virtual model as you work through the design process.

Categories

The basic set of object categories can be viewed and edited in the following way:

1. On the menu bar, click Settings ➢ Object Styles, or on the menu bar click View ➢ Visibility/Graphics.
2. In the resulting dialog box, click the Object Styles button at the bottom.

Figure 1.16 shows the Object Styles dialog box. Checking the Show Categories from All Disciplines check box will allow you to see additional categories primarily used by the Revit Architecture and Revit MEP platforms. This dialog box organizes the categories into three tabs: Model Objects, Annotation Objects, and Imported Objects.

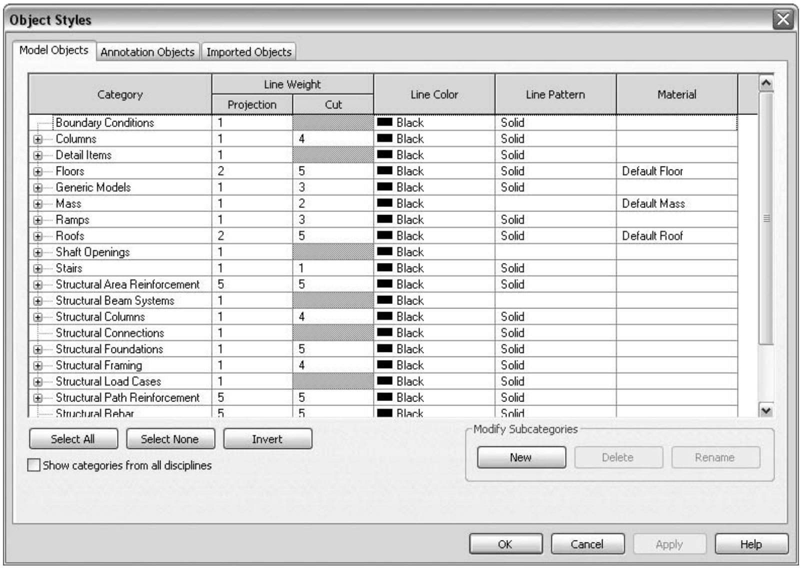


FIGURE 1.16
The Object Styles dialog box provides display controls for element categories.

Families

The object categories are divided into *families*. Families can be system, in-place, or external. System families are built into templates and projects, and in most cases cannot be completely removed. System families can be:

- Model elements (walls, columns)
- Datum elements (grids and levels)
- View-specific elements (text notes, tags)

System families are easily identified by checking the family name of an element in the Element Properties dialog box. The name of the family is prefaced by System Family (see Figure 1.17).

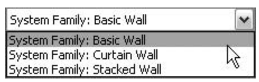


FIGURE 1.17
The name of the family is prefaced by System Family.

External families are loaded from the Revit Structure content libraries as well as from your own custom created libraries. You load them in as you need them. These family files can be created from scratch using a default template shipped with the program, by using a custom template, or by copying and modifying a similar existing family.

Use the Built-In Libraries TO Your Advantage

Copying and modifying the built-in library families is a great way to learn how families are created, and to begin experimenting with your own adaptations. Learning to adapt and create families will prove to be a big benefit for you.

External families can be:

- Model elements (columns and beams)
- View-specific elements (tags and detail components)

In-place families are primarily used for custom project-specific applications. As the name suggests, these families are built in-place within the project. One possible application for an in-place family would be a void family used to cut an odd-shaped opening in a wall.

Types

Families are further divided into types. A family may have one or more types. A column family (model element) type would be a W12x40. A grid family (datum element) type would be a ¼″ Bubble. A structural framing tag family (view-specific element) type would be Boxed.

Instances

An instance is a specific element of a given family type. For example, for a model element of the Category Structural Column, whose Family is a W-Wide Flange Column and whose Type is a W12x40, the instance would be the specific W12x40 located at the grid intersection A-1 from Level 1 to 4′-0″ above Level 2.

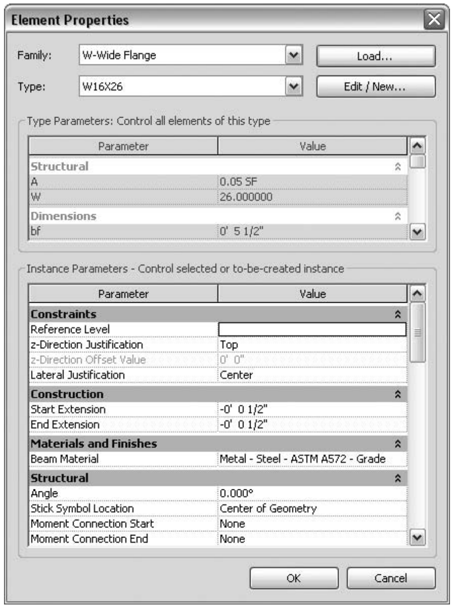
Element Properties Dialog Box

As you are placing objects such as columns and beams into your building model, you can access the properties of each one by highlighting it and clicking the Properties button on the Options bar (next to the Type Selector), which will open the Element Properties dialog box (see Figure 1.18).

Elements can also be accessed in other ways. With the element(s) selected:

- Right-click, then select Element Properties.
- From the menu bar, choose Edit ➢ Properties.
- Press Alt+Enter on your keyboard.
- Press the default shortcut PR.

FIGURE 1.18
Element Properties dialog box



This dialog box contains the parameters that define the element, and is broken into two main areas: instance and type parameters. It can be used to change their values in the following ways:

- Changes made to the instance parameters will only affect the element(s) that you have selected.
- Changes made to the type parameters affect every instance of the element that you have in the model whether you have selected it or not.

Now that you have studied the various elements that are available to you for modeling your structure, let's examine how those elements are displayed in your model. The views that you create, their interaction, and how they display the elements in the model are important considerations in the evolution of any project.

Project Views and Display

There are numerous types of views that are created as you are working. Each View type serves a variety of purposes within your project. You access these views through the Project Browser and display them in the drawing area. In this section, we will introduce you to these various view types with a brief explanation of how they work.

- Plan, elevation, callout, section, and 3D views act as direct graphical "windows" to the model.
- View-specific elements such as notes and member tags can then be placed and the graphical display of elements can be manipulated in these views without any effect on the actual model or other views. Modifications can be made directly to the model elements in any of

these views, and those modifications will be instantaneously propagated to all other relevant views (as you will recall, this is called bidirectional associativity).

- Drafting views are somewhat detached from the model itself, but they play a valuable role in the development of construction documents.
- Sheet views are a specialized view that typically contains one or more other view types and your title block. They are used for creating your finished construction documents and presentation drawings.

Now let's look at each one of these view types and see how they can be created in your project.

Plans

One thing to understand about plan views is their relationship to the levels you create. A new plan view is not automatically added to the Project Browser every time you draw a new level in your project. Copying an existing level to create a new one will not automatically create a plan view. In these cases, you must create the view after the level is created.

To create a new plan view for a level that does not currently have one, follow these steps:

1. Select the Floor Plan command from the View tab of the Design bar, or from the menu bar select View ➤ New ➤ Floor Plan.
2. Select one or more levels to create plan views in the New Plan dialog box (see Figure 1.19).

If you are using this method to create a duplicate plan, be sure the Do Not Duplicate Existing Views check box is not selected. Also note that you can preset the scale of the plans being created.

FIGURE 1.19
New Plan dialog box

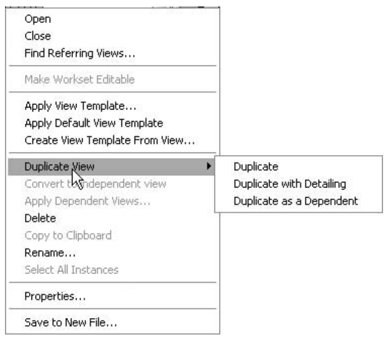
You can duplicate plans by right-clicking a plan view name in the Project Browser and selecting Duplicate View from the context menu. A fly-out menu (see Figure 1.20) will appear with three options:

- Duplicate
- Duplicate with Detailing
- Duplicate as a Dependent

These three ways to duplicate views work like this:

- Duplicate will create a new plan view that is an exact copy, displaying all datum and model elements. No view-specific elements (such as text) will be copied with this option.
- Duplicate with Detailing works the same way except that all view-specific elements are also copied.
- Duplicate as a Dependent will create a child view to the selected parent view. Any number of dependent views can be created from a single parent view. This feature was added to facilitate the division of large overall plan views into smaller views for placement on sheets. All view-specific elements are shared between the parent and all child views.

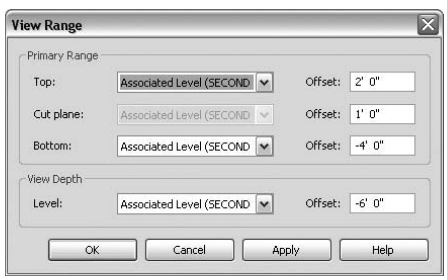
FIGURE 1.20
You can use Duplicate View to create copies of a view.



View Range

View Range is important to understand when you are dealing with plan views since it controls the basic vertical range above and below the elevation that your view is cut through the model. You access the View Range properties by clicking View Range in the View Properties dialog box of that particular plan view. The resulting View Range dialog box (see Figure 1.21) is used to control element visibility and display that is perpendicular to the plan view.

FIGURE 1.21
View Range dialog box



The primary range is defined by three horizontal planes: top, cut, and bottom. A fourth plane that can exist outside the primary range is the view depth. The cut plane is always defined as an offset of the view's associated level. The other three planes may be defined relative to any level in the model or set to Unlimited. The top and bottom planes of the view range define the primary vertical extents of the model that is displayed in the view:

- Model elements that fall within the primary range and that are not cut by the cut plane will display their projection line style as configured in the Visibility Graphics dialog box.
- Elements that are cut by the cut plane will display their cut line style (if they have one), as configured in the Visibility Graphics dialog box.

Cut Lines and Patterns

Whether or not a cut line or pattern style exists for a particular category can be verified in the Object Styles or Visibility/Graphic Overrides dialog box (see Figure 1.22 later in this chapter).

Categories that have a shaded cell in the Cut-Lines column do not have a definable cut line style.

Elements that are outside of the primary range but fall within the extents of the view depth will be displayed with the Beyond line style. The Beyond line style is defined in the Line Styles dialog box, which you open by choosing Settings ➢ Line Styles from the menu bar.

You might wonder where you would use the Beyond line style. The most obvious example is an architectural roof plan. Usually the architectural plan displays the roofs looking down from above the building on all the roof levels below. You can have a roof at the second floor and a roof at the sixth floor showing in one plan view. To do that, you extend the view depth down to the second floor so that all the roofs are within the range, and you can give it a specific line style to distinguish it as being beyond the view cut plane.

Real World Scenario

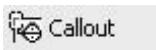
Multiple-View Concept

Although it may not seem natural at first, a good practice to adopt while working with views in Revit Structure is to create multiple copies of the same view to be used for specific purposes. When dealing with plan views, you will typically have at least three views for each level.

Using the second level of the model as an example, you can create the following views: S-FP02, S-FP02-Analytical, and S-FP02-Working. The S-FP02 view would be placed on a sheet view and will ultimately be plotted as part of the construction documents. The graphical displays of the datum and model elements in this view are always set as they are intended to plot. Only construction document view–specific elements are added to this view. The S-FP02-Analytical view is used to display an analytical view of the model elements. S-FP02-Working is typically used to coordinate with the other disciplines by linking in models and DWG backgrounds or to adjust the display of elements to troubleshoot the model. Annotations can be placed in either of these views as reminders or notes to other team members.

Working in this manner helps safeguard the integrity of the construction documents without hindering productivity.

Callouts



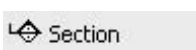
Callout views are used to produce a blow-up of an area for clarification and can be accessed on the Design bar on the View tab. This usually larger-scale view is used to show a higher level of detail and additional annotation that may not be legible at the original scale.

Three distinct types of callouts are available: reference, detail, and view.

Reference callouts are ideal for tying in standard details (drafting views) or to refer to a similar existing view. Reference callouts do not create a view in the project; instead they are tied directly to the existing view they reference. You place reference views by selecting the callout command from the View tab and selecting the Reference Other View box on the Options bar. Then, select a view to reference from the drop-down menu on the Options bar. Finish by drawing the callout graphic in a view.

You place detail and view callouts by selecting the callout command from View tab of the Design bar. Then select the appropriate callout type from the Type Selector, and finish by drawing the callout graphic in a view. Detail callouts will place a new view under the Detail Views (Detail) heading in the Project Browser. Detail views are typically used to embellish another detail or section at a larger scale. View callouts will place a new view under the same heading as the view it was created in. View callouts have all of the same capabilities as the view to which they refer. This makes them ideal candidates for enlarged plans.

Sections



Section views cut vertically through the model. These views are created for many purposes, but mostly for wall and building sections. The section updates automatically with any new or modified model elements falling within its scope (that is, its length and depth). Other sections that fall within that scope are also displayed by default. That makes referencing mistakes much less likely to occur, and saves you a lot of valuable time that you might otherwise spend cross-referencing details.

During the course of modeling or troubleshooting, it is often useful to cut a working section. These working views are not intended to be placed on the construction documents and are essentially disposable. To keep the Project Browser organized and the construction documents free of “view clutter,” it’s a good idea to create a new section view type that is easily identifiable. You need to be careful not to move sections that you have placed on sheets.

Sections views that that have not been dragged onto a sheet and thus are not referenced on any sheet will not print, so making working sections and leaving them at print time is fine. But if you export your model to an AutoCAD DWG file, the unreferenced section callouts will be exported, so you will have to do a little cleanup of the DWG and erase them before you send it out, especially if it is an important design submission. So it is better to erase those sections before you export your sheets.

To add a section, perform these steps:

1. Click Section on the View tab.
2. Click on the location you wish to begin the section.
3. Click where you want to finish the section.
4. Highlight the section, which will display a dashed green line.
5. Click the grips and drag the dashed green line line box to adjust the depth of the section.

Elevations

Two types of elevations are available in Revit Structure: building and framing.

You create building elevations by selecting the Elevation command from the View tab of the Design bar and placing the view tag in a plan view.

Building elevations are similar to section views but are located under a separate heading in the Project Browser. Each instance is capable of generating the four directional views.

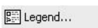
Framing elevations are specialized elevations designed to facilitate the placement of vertical bracing and moment frames. You create these elevations by selecting the Framing Elevation command from the View tab of the Design bar and selecting a grid or named reference plane in a plan view to attach the elevation view tag. You can place the elevation view tag on either side of the grid or named reference plane; it depends on which side of the element you are prior to placement.

Drafting

Drafting views are 2D views that have no connection to the 3D model. To work in the drafting views, you will use the drafting tools found on the Drafting tab of the Design bar, such as Detail Lines and Filled Regions.

A good example of a drafting view is a typical detail that you wish to add, or an imported scan. Detail views can be saved and inserted from detail libraries either individually or as whole sheets, as you will see later in Chapter 10.

Legends



Legends are unique views which have the advantage that they are the only view types that can be placed on multiple sheets. You create a legend by selecting the Legend command from the View tab of the Design bar. Legends views are typically used as an explanatory list of symbols and text that are found in the project. Chapter 10 will discuss alternative uses for legend views and step you through the creation of a legend.

Schedules

With the exception of the Graphical Column Schedule, schedules are typically spreadsheet-style, text-based views that report the quantities of specific elements or the values of an element's parameters. You create a schedule by selecting either the Schedule/Quantities or the Graphical Column Schedule command from the View tab of the Design bar.

Creating schedules will be discussed in depth in Chapter 11.

3D

Three-dimensional (3D) views allow you to orbit around your model in 3D space in order to view the overall model. You can activate a section box that you use to create cutaway views through any portion of the overall model. 3D views are very important for visualization purposes, both for the client and the working engineer or draftsperson. When you start your project it is good practice to immediately create a sheet of 3D views to share with your client. It immediately creates a better understanding of the structure, especially for an owner who has no experience with reading plans and sections.

Sheets

Sheet views contain your title blocks, and they are the assembly point for all the different views you create: plans, sections, elevations, and so forth. You drag the different views you have created onto the title sheet and position them as necessary. When you add a sheet view to your project, you will be prompted to select a title block to use for the new sheet. The title block itself is a separate file that you load into your project. You will learn more about title sheet creation in Chapter 12.

Plan Region

Plan regions are self-contained view ranges within a plan view and are used in those cases where you need the view range of a particular area on a plan to differ from the overall plan view range. Remember that the view range sets the vertical range above and below the elevation at which your plan view is cut. An example of this is an area with a stepped footing that is stepping out of the vertical range. If you want to control exactly which step to display on the overall plan, you use a plan region to adjust where the stepped footing cuts off in the view.

When you start the command you are put into Sketch mode. To create a plan region, simply sketch the area you wish it to affect. Then adjust the view range parameter in the View Properties dialog box to suit your needs.

Visibility/Graphic Overrides Dialog Box

The Visibility/Graphic Overrides dialog box (see Figure 1.22) provides a way to change the display of elements for a specific view.

The changes made in this dialog box have no effect on the model itself or any other views, including any dependent views and the current view. If your intention is to change the look of elements project wide, see the appropriate dialog boxes outlined in the upcoming section, "Graphic Standards." Line style settings are located in the Visibility/Graphics Overrides dialog box. For easy access, press VG to access these settings, as you will be in and out of this dialog box constantly.

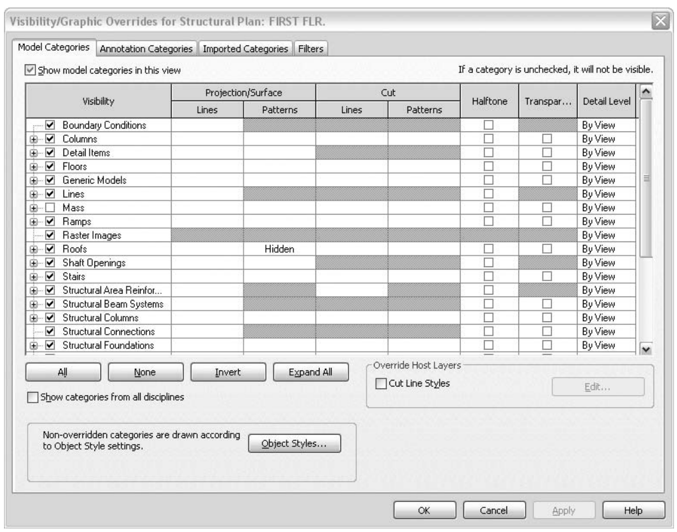
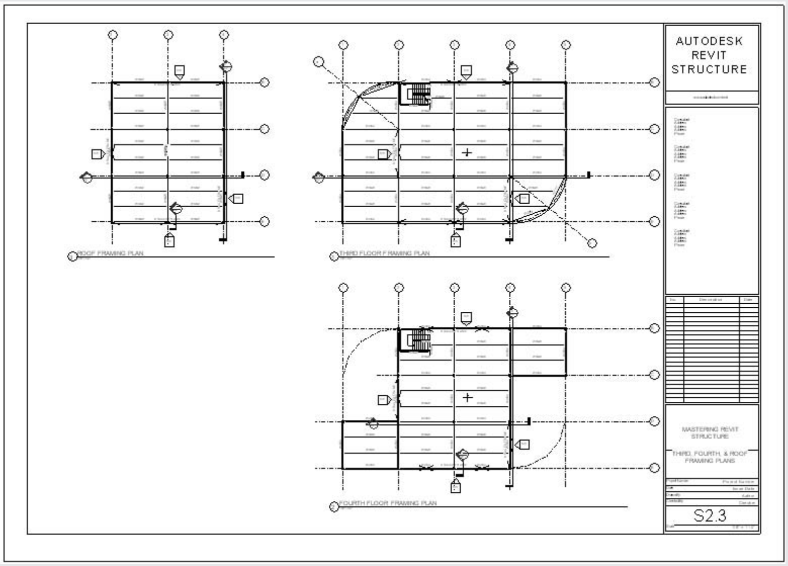


FIGURE 1.22
Visibility/Graphic Overrides dialog box

The Importance of Display Control

Remember that display control is half the effort that must be expended toward the goal of getting a good-looking set of construction documents from your model, like the sheet of plans in the graphic below. It is easy to get caught up with building the model and neglect setting up and creating the documents that must be derived from it. At the beginning of the project, start thinking about how you will organize your different views onto sheets and how you want each view displayed. Set up your title block and start adding sheets as soon as possible. You should be asking yourself questions such as what detail mode and view scale is appropriate, and how much of the model should a certain view expose.

Getting your modeled elements to "look right" on the final document can be a frustrating task and requires special attention to the methods that will be discussed. So keep your deadlines in mind and schedule your time accordingly so that you are not only modeling but also creating your documents in a wise manner.



Now that you have studied the various views that are available to you for displaying your virtual structure, we'll show you how to adjust the graphical settings within those views to best display the elements in your final documents.

Graphic Standards

What if you want increase the cut line width of your structural columns and have it change in every view of the project, not just in the current view?

The graphical display of elements throughout all views of a project is efficiently managed with a handful of dialog boxes. This dynamic ability to modify project standards on the fly during production is a major benefit of the Revit Structure platform. You will now explore the different ways that you can effectively configure and control your element displays.

Object Styles

One of the most important of these dialog boxes is the Object Styes dialog box. In the Object Styles dialog box, you assign and control the following:

- Line weights (both projection and cut)
- Line colors
- Line patterns
- Material styles

This dialog box controls the display of all categories and subcategories of elements in the project model. You organize and modify these styles using the Object Styles dialog box (see Figure 1.23), which you access by clicking Settings ➢ Object Styles or via the Visibility/Graphic Overrides dialog box. There you can access and edit the various display parameters, such as cut line and projected line weights.

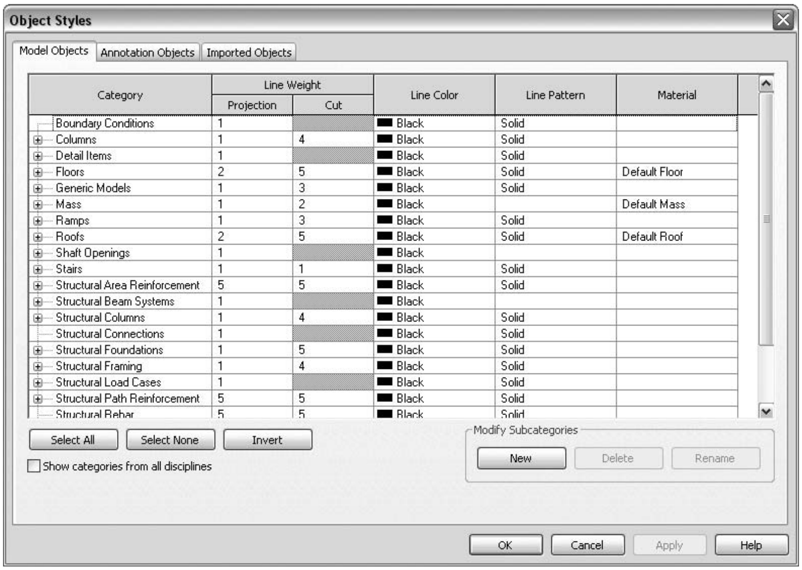


FIGURE 1.23
Object Styles dialog box

Material Styles

The Materials dialog box allows you to configure and control all the material assignments for elements in your model. For instance, what grade of steel do you want your columns to be and in what color do you want it displayed in a 3D view?

Open the Materials dialog box (see Figure 1.24) by choosing Settings ➢ Materials from the menu bar. This dialog box lists all of the materials currently available in the project:

- The Graphics tab defines the appearance of the material in all nonrendered views.
- The Render Appearance tab, as the name suggests, contains settings that define how the materials will appear when rendered.

- The Identity tab contains parameters that can be specified for each material that can be leveraged by tags and schedules.
- The Physical tab contains structural information that can be leveraged for the structural analysis of the model.

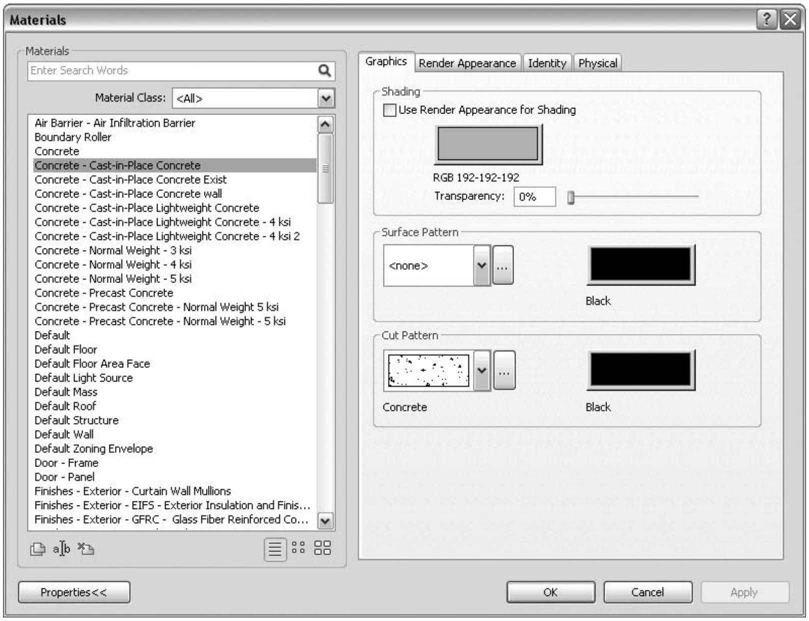


FIGURE 1.24 Materials dialog box

Line Styles

Line Styles are used mostly for 2D drafting and for using the Linework tool. There are many default styles. Nine of those cannot be deleted. Beyond that you can create your own line styles.

Open the Line Styles dialog box (see Figure 1.25) by choosing Settings ➢ Line Styles from the menu bar. This dialog box lets you view and edit all of the line styles available in the project.

Line Weights

You define line weights using the Line Weights dialog box (see Figure 1.26). Open this dialog box by choosing Settings ➢ Line Weights from the menu bar. Line weights are divided into three categories: Model, Perspective, and Annotation. Sixteen line weights can be defined for each of these categories. The Model category can have unique weights for each of the predefined 28 imperial or 12 metric scales. Weights defined for the perspective and annotation categories are absolute, regardless of scale.

FIGURE 1.25
Line Styles dialog box

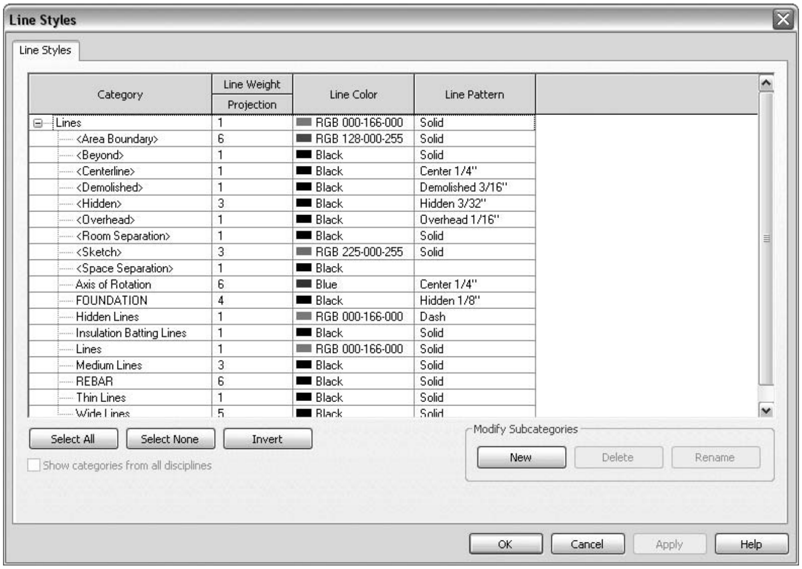
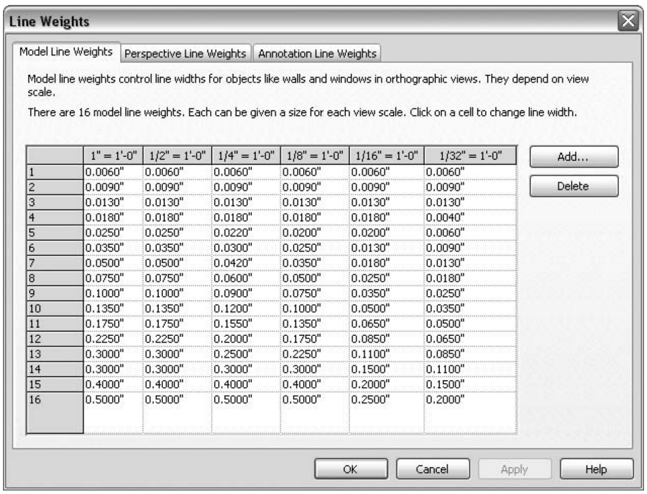


FIGURE 1.26
Line Weights dialog box



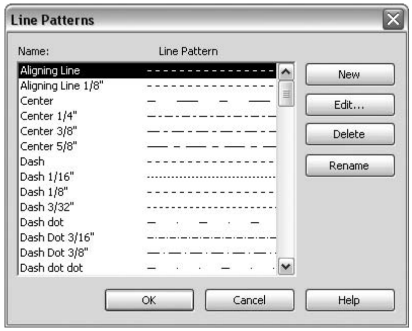
LINE WEIGHT ADJUSTMENTS

Different plotters may require adjustment of the default line weights in order to get a better print. In some cases, the thinnest pen weights will not display well on certain plotters, so you should experiment with different values to see which are best for your particular output device.

Line Patterns

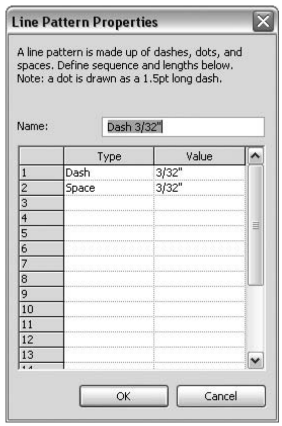
Line patterns are a sequence of dots, dashes, and spaces of various lengths that create distinct lines. The Line Patterns dialog box (see Figure 1.27) can be accessed from the menu bar; simply choose Settings ➢ Line Patterns.

FIGURE 1.27
Line Patterns dialog box



Clicking the New or Edit button in this dialog box will take you to the Line Pattern Properties dialog box (see Figure 1.28). Here you can develop new line patterns or adjust existing ones in order to perfect the different view displays.

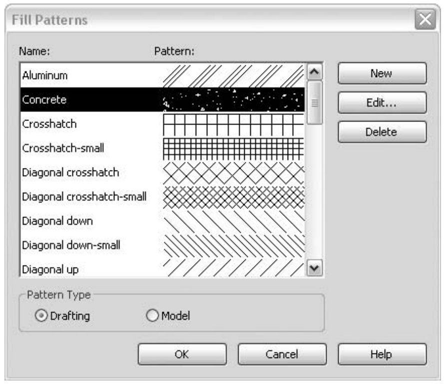
FIGURE 1.28
Line Pattern Properties dialog box



Fill Patterns

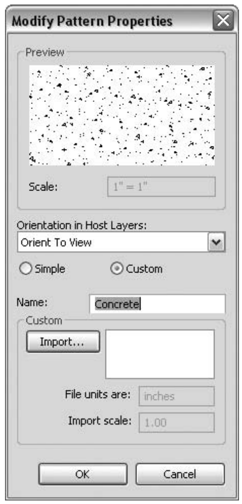
Fill patterns are used by filled regions for hatching. They are also used by materials for both surface and cut patterns of objects. All of the fill patterns can be viewed and edited in the Fill Patterns dialog box (see Figure 1.29). To open this dialog box, choose Settings ➢ Fill Patterns from the menu bar. Revit Structure offers two types of fill patterns: drafting and model. Model elements scale with the object and represent the real-world appearance of an object, like CMU (concrete masonry units) coursing. Drafting patterns do not scale and are a symbolic representation of a material such as the concrete pattern.

FIGURE 1.29
Fill Patterns dialog box



Clicking the New or Edit button in this dialog box which will take you to the Modify Pattern Properties dialog box (see Figure 1.30). Here you can develop new patterns or modify existing ones.

FIGURE 1.30
Modify Pattern Properties dialog box



That completes your first look at the Revit Structure interface and its basic set of commands. In the next chapter you will learn how to develop different project environments through the use of templates. We will discuss the various settings that are available to you as well as how the Project Browser can be organized so it can be managed. In the end we will discuss how content and settings can be transferred from one project to another.

The Bottom Line

Use the graphical user interface. The Revit Structure GUI is an easy way to interact with your computer in order to efficiently create your project model and documents.

Master It There are several ways to launch a single command in Revit Structure. List the various ways in which the Beam command can be invoked. Which method is the quickest?

Understand the types of elements in the modeling environment. In the modeling environment, there are basic types of model and annotation elements that you use in the construction of the virtual model and construction documents that you derive from the model.

Master It Modeled elements have a defined hierarchy that consists of categories, families, types, and instances. Use a structural column and give examples of each of these four element properties.

Create and manage project views. Even though you are building a 3D model, most of the time you are working in 2D views such as plans and sections. Therefore, the view types become your working planes and must be sensibly arranged.

Master It List all the major project view types discussed in this chapter.

Control the graphical display of elements in a project. Creating the model is only half the story. Then you must derive the 2D and 3D views you will need for your construction documents. Each of these views must be able to display the model, and those display controls are an essential subject to understand.

Master It In your project you want to change the look of your masonry units on plans and elevations to match your company standards. Explain how to change the cut pattern for Concrete Masonry units to a diagonal pattern and the surface pattern to 8×8 block.

Chapter 2

Setting the Project Environment

To fully master Revit Structure, you need to understand the settings that are available to you and distinguish between settings that are global for the project or specific to the user's machine. Properly using these settings will easily help you achieve the look you want for your structural documentation. Most of the settings that will be discussed in this chapter will become part of your standards and should be incorporated into template files.

Getting a handle on the settings that are part of the project environment, setting them appropriately for your company's best practices, and making use of well-thought-out templates will help you start your projects out much faster. This will allow users to focus more on the project itself rather than the behind-the-scenes items that will become your standards.

To take full advantage of Revit Structure's organizational methods, you will learn how to make use of browser organization and naming conventions to aid in the modeling process, and how to create a clutter-free environment. Using custom templates to start or populate your project will help you follow standards, and the documenting efforts for your model will be more efficient.

Making these settings and providing proper organizational workflows will force users to maintain standards, thus reducing errors and keeping users working efficiently.

In this chapter you will learn to:

- Develop your own custom templates
- Set project units and precision display
- Make adjustments to structural-specific settings
- Organize the Project Browser
- Transfer standards into your project

Working with Project Templates

Project templates offer the initial conditions for you when your project is started. They can also be useful to help put content into your project as you reach different stages of the modeling process. You can think of a template as a preset starting point. Templates have an `.rte` file extension rather than the`.rvt` extension that a project uses. As you create your standards and develop your working environment, be thinking about where and how that information will fit into your project template(s). Taking the time to set up your templates effectively and update them continually as the software improves or your standards change will help eliminate the redundant work of constantly making changes to your project settings. Not only that, but a properly tuned template will help maintain standards throughout your office.

It is important to know that when setting up your template(s), you definitely will not get everything you need in them the first time. Templates are something that will more than likely change with each and every project you do. You will continue to incorporate new content as you learn the software and to reflect your workflow. Eventually you will find that the changes will be reduced, but as new releases of the software become available, you will have to adjust your templates to accommodate any new changes that you want to bring into your workflow.

The Ingredients of a Good Template

Templates can contain just about everything that a normal project would contain. They can store project information, project settings, line styles, line weights, project views that have predefined settings, and visibility/graphics settings. If you have projects that all start out with similar geometry, you could put that geometry into the template and use it to start all your projects that use that geometry. Using this approach, you won't have to model elements over and over again.

If you are transitioning from an AutoCAD environment, you probably want to take all those years that went into the look and feel of the documentation that your company has established and continue that look in Revit Structure. All this can be done, but you'll have to modify what comes out of the box. Revit Structure allows you to easily adapt your own look and feel, which will be discussed throughout this book in other chapters. However, when you do develop these standards, you need to place them in templates so they can easily be applied to projects to help keep a constant look and facilitate users following standards. Here is a list of some factors to keep in mind when developing your template files:

The display of graphics

- Fill patterns (hatching)
- Materials
- Object styles
- Line styles
- Line weights
- Line patterns
- Structural symbols

The display of annotation

- Text styles
- Dimension styles
- View tags
- Annotation tags

Project settings

- Project units
- Structural settings
- Rebar settings

This list does not contain everything, but includes critical aspects that you will want to address prior to starting a project.

As your understanding of using Revit Structure progresses and becomes clearer, you will find yourself putting more information into your templates and even creating several different templates. You may have a template set up for projects with different structural systems. For example, a project for a precast parking garage may not use steel members of any kind, so these families would not be loaded in that template. If your company works on a lot of parking garages where the structural engineer drives the project design, your template could already contain preset views, levels, and maybe even grids, and you will just have to adjust their location.

If you have created schedules such as a Footing Schedule, Pier Schedule, Sheet Index, or schedules that allow you to do quantity take-offs, those schedules could already be created inside your templates. Users do not have to keep remaking these schedules. Not only does this approach save you time, but it also forces all schedules of different makes and types to have a consistent look.

As you become more familiar with Revit Structure and learn about the objects and the types you'll be using, you can also add the following information to your template(s):

- Wall types
- Wall footing types
- Foundation slab types
- Slab types
- Roof types
- 2d components
- Families
- Analysis loading information
- Schedules
- View templates
- Sheet setup

No Worksets Allowed

All commands that are used in a worksharing environment will be grayed out in a template file. Worksets cannot be included in a template.

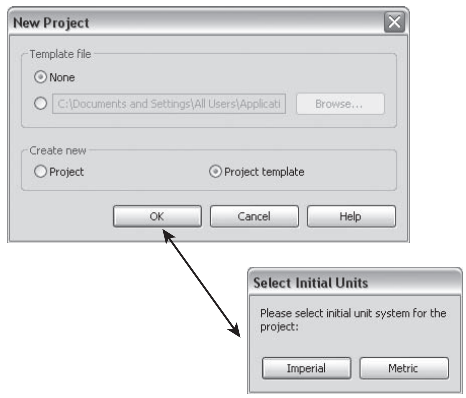
Many of these items can be transferred from your projects into a bare template or project that you can create from the None template option when starting. Not only will this allow you to create content for your templates as you work on projects, but putting this content into an empty project first will let you clean out any non-project-specific data before placing it into your template.

Using the No Template Option

The No Template option will start you out with the bare minimum of a model. It will have one structural plan view to start the modeling process, and no families will be loaded. The only families available will be system families, and they will contain only one generic type.

This template should not be used to start your projects, but it will come in handy when you want to build templates or project files to quickly add content to your project without bringing over numerous amounts of other settings. For instance, say you wanted to create a template with several wall types for concrete; you would start this process by choosing File ➢ New ➢ Project. Figure 2.1 shows the resulting dialog box, where you will create a new project. In this case, you will create a new project template (which will use the `.rte` file extension) and start it by not using a template. When you specify None for a template, you will also be presented with an option to use Imperial or Metric for your units. This is also true when you're creating a project from a None template. Once your blank template is started, you can create only the wall types that you want as part of your concrete wall types. Save the template in your Imperial or Metric Template folder, and use it to transfer into your active project model when required (see the section "Transferring Project Standards" later in this chapter).

FIGURE 2.1
Starting a project template from a None template



It is important to know what to put into your templates and know that you will not be able to create the perfect template right out of the starting gate. Ensuring that your templates are current with your standards prior to starting each project or investing the time up front to create a template for a particular project can prevent inconsistencies as well as increase productivity downstream.

Creating a New Custom Project Template

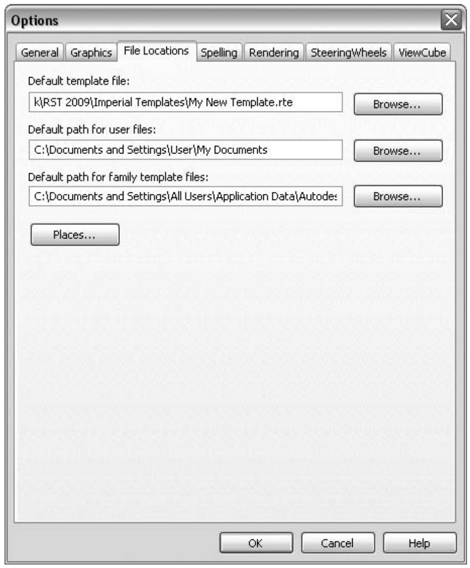
You can create a new template in several ways:

- Start from Revit Structure's default Template `Structural Analysis-default.rte`.
- Start from any other existing template.
- Save a current project (RVT) as a template.

If you are just starting to use Revit Structure, a good approach is to start a new template from Revit Structure's default template `Structural Analysis-default.rte`. Add, remove, or revise existing settings however you see fit for your office environment and to achieve the look and feel that you want for your documentation. Once you have completed this template, use it to start your projects. As you continue to work on and plot your projects, you will find that certain settings will need to be tweaked. Continue to make these adjustments within your active project but not on your template. At any time during the project, you can either transfer your settings from your project into your template or save your project as a template, and then delete unnecessary model objects and views as required.

Your project templates should be stored on your network with all your other custom content so other users can access them. If you have a general template that will be used, you can set it as the default template to be used whenever someone starts a new project. As Figure 2.2 shows, you can do this by choosing Settings ➤ Options and clicking the File Location tab. Under Default Template File, click the Browse button and select the path to your template file. When a new project is started, this is the template that it will use unless you browse to a different file.

FIGURE 2.2
Setting the default template location



Templates should be saved to the network for all users to access and named so that users will easily know what is inside them as well as when to use them.

Starting a Project from a Template

Once you have your templates ready, you can start your new project by choosing File ➤ New ➤ Project or by going to Revit Structure's Recent File window, shown in Figure 2.3. This window is usually displayed when Revit Structure starts. If not, it can be toggled on by choosing Windows ➤ Recent Files.

FIGURE 2.3
Use the Recent File window to start a new project or family, or to open an existing project.

When you click New to create a project, the New Project dialog box shown in Figure 2.4 opens.

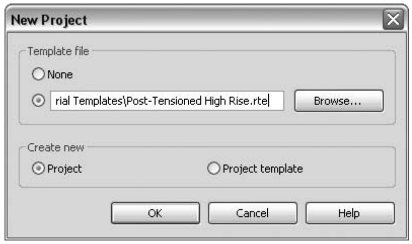


FIGURE 2.4
Starting a new project from a custom template file

You will have two options to specify how you want to start out. The first one deals with the template file. You can choose not to use a template or to use one of your custom-made template files. In this example, the template that contains all the information for a high-rise post-tensioned structure will be used.

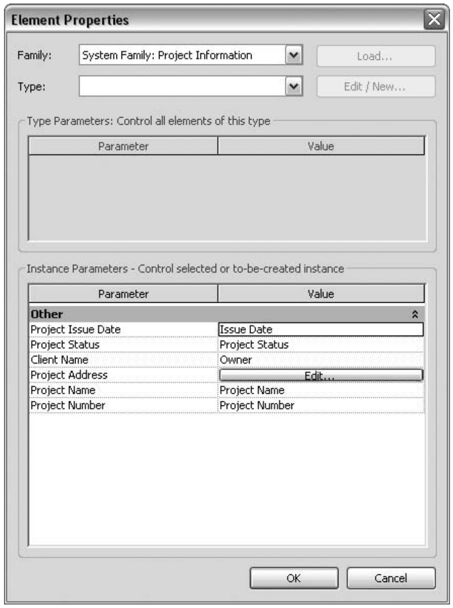
Your second option will be whether your new file will be a project or a project template. In this case, selecting Create New – Project will preload your new project with all the settings stored in the template file you have chosen. Your project is under way and you are ready to start adding content. You have not named it yet, so don't forget to give your project a name and save it to your network.

Project Information

Project information is data pertaining to the project that typically does not change. You can find this category in Revit Structure by selecting Settings ➢ Project Information to open the dialog box shown in Figure 2.5. This information is typically shown in title blocks, and since it is specific to the project, all parameters are considered type parameters.

The initial six parameters that are available in this box are hardcoded into Revit Structure and cannot be changed. You are allowed to add parameters to this category by adding project parameters, which are parameters you can add in your project and that can be globally assigned to several categories instead of inside families.

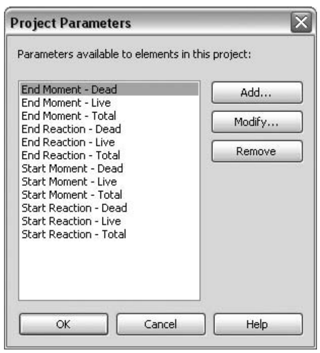
FIGURE 2.5
The Project Information Element Properties dialog box



To add a parameter for contractor information, follow these steps:

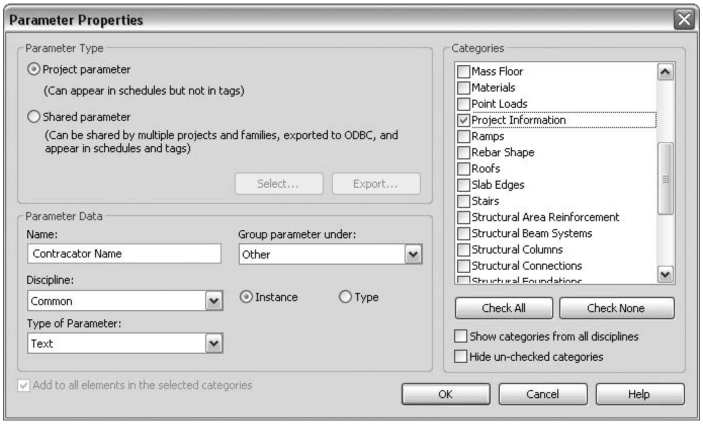
1. While in your project, choose Settings ➢ Project Parameters. Figure 2.6 shows the Project Parameters dialog box.

FIGURE 2.6
The Project Parameters dialog box



2. Click Add and fill in the information in the Parameter Properties dialog box shown in Figure 2.7. In this case, you will be adding a project parameter to the category Project Information with a name of Contractor Name. Its type will be Text and it will appear under the Other tab.

FIGURE 2.7
Adding a project parameter to the Project Information category

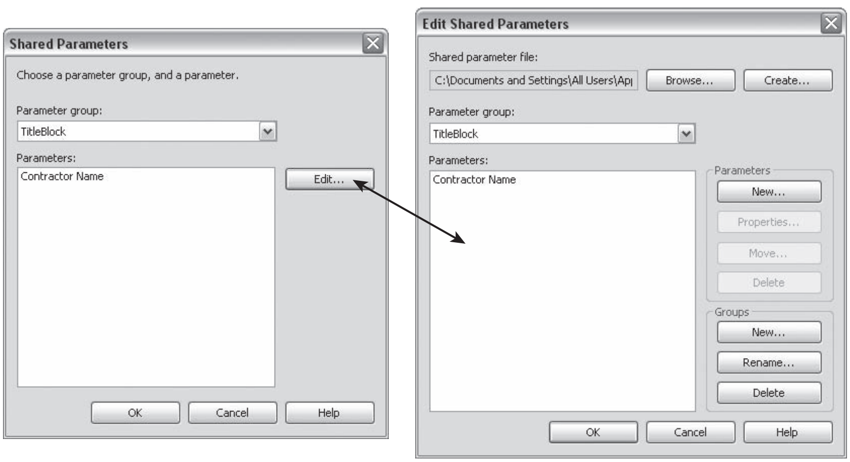


Realize that just adding this new parameter as a project parameter (not a shared parameter) doesn't do much for you other than having the information shown when you display the Project Information Element Properties dialog box.

Adding Project Information to a Title Block

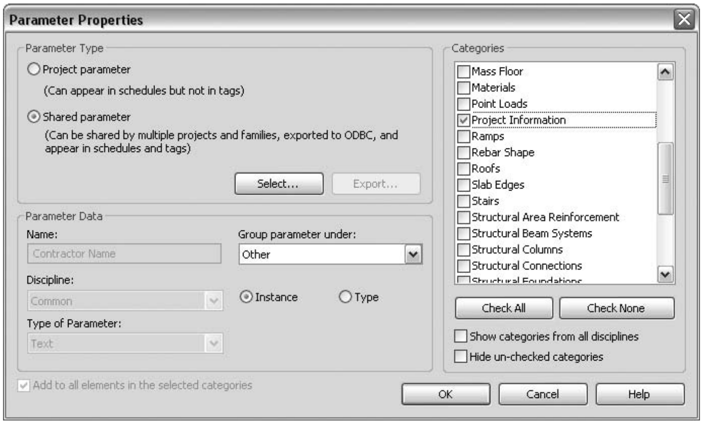
If you want to see this information in a title block, you will have to add it as a shared parameter. Shared parameters are parameters that you can add to families or projects. They are stored in a separate text file where they can be accessed and shared in other families or projects. Once you select the Shared Parameter option, you will have to select a parameter from a list. If the parameter is not there, you will have to create it by clicking the Edit button (see Figure 2.8).

FIGURE 2.8
You can either select an existing parameter or create a new one on the fly.



As Figure 2.9 shows, the Parameter Data options in the Parameter Properties box will be grayed out. This is because this information is stored inside the shared parameter.

FIGURE 2.9
Adding a shared parameter to the Project Information category

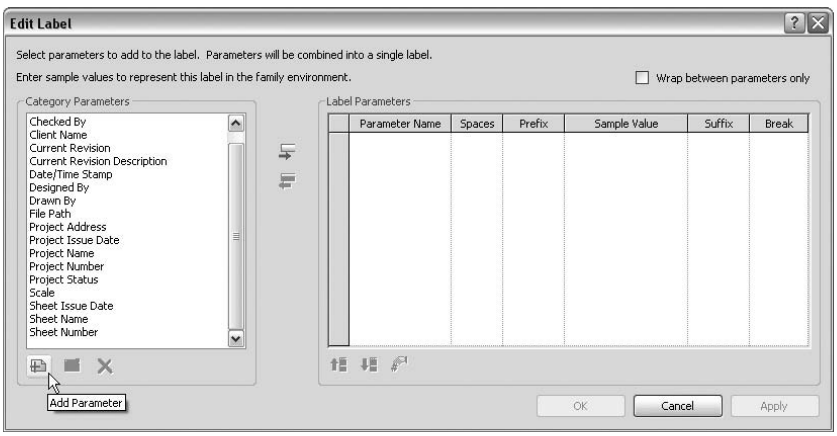


HEY, WHERE DID THE CATEGORY GO?

Typically, project information acts as a type parameter. In this case, setting the Shared Parameter option to Type causes the Project Information category to disappear from the list. You only need to set it to Instance and Revit Structure will take care of the rest. This instance parameter will automatically act as a type parameter.

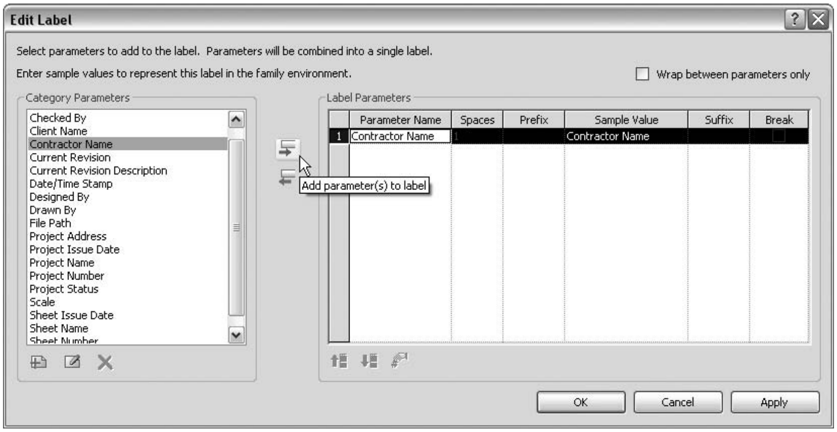
Inside the title block you will have to add the same shared parameter to a text label. Do this by placing a new label or by selecting an existing label and editing it using the Options bar. Figure 2.10 shows the Edit Label dialog box. When adding the parameter at this stage, you are only allowed to add a shared parameter. These steps are similar to those for adding a shared parameter in the project.

FIGURE 2.10
Adding a shared parameter to a label



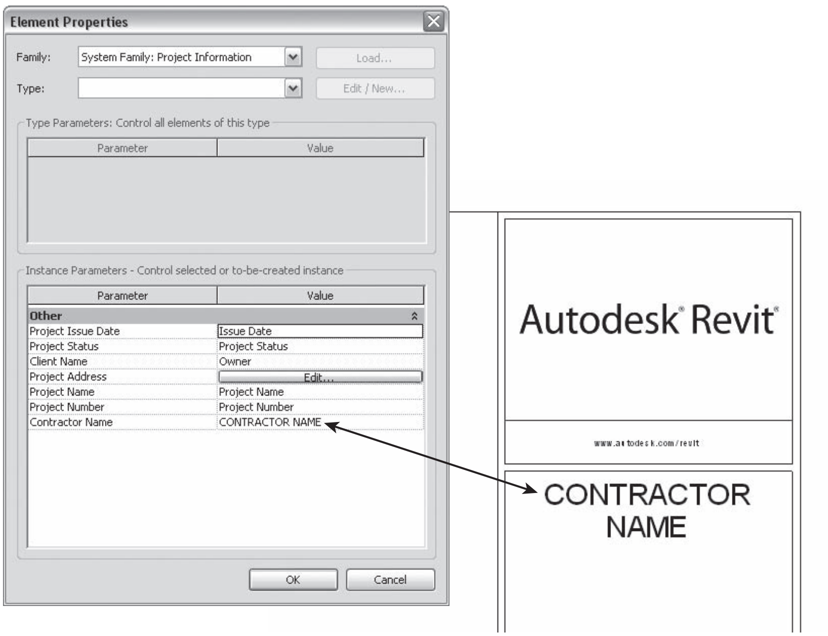
You will still need to add the shared parameter to the label by selecting the parameter in the left panel and clicking the right-arrow icon to move it over to the right panel, as shown in Figure 2.11.

FIGURE 2.11
Putting parameters into a label



When the title block is loaded into your project with that label, it will link itself to the Project Information shared parameter you created. This will allow you to change the Contractor Name information through the title block or through the Project Information dialog box. Figure 2.12 shows the information linked to the title block.

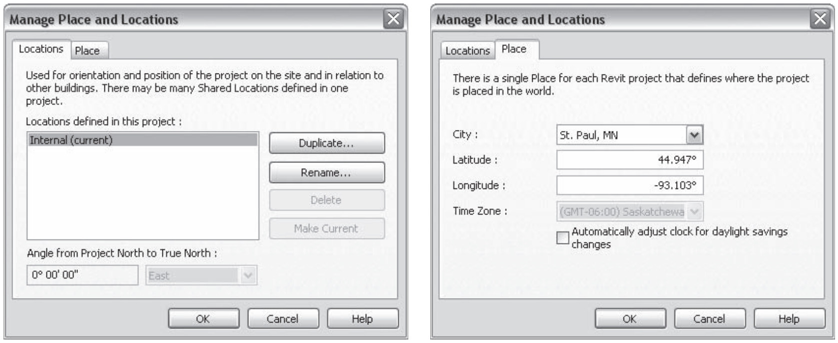
FIGURE 2.12
User-added project information linked to a title block



Project Location

Revit Structure acknowledges that every project needs a location with regard to where it is placed in the coordinate system. You can find the location of this information by choosing Settings ➢ Manage Place and Locations. Figure 2.13 shows the information that can be recorded and kept as part of your project information.

FIGURE 2.13
The Manage Place and Locations dialog box



You will usually place your first object somewhere in the middle of your drawing area in that big white space (it will be black space if your background is set to black). If you are starting your project from an architectural model, you are probably going to link it into your project and use the same origin point as the architect did. In Revit Structure, you not only have a location in the left/right or up/down direction, but you have a Z direction as well. Is your first floor at elevation 0´-0˝ or is it at 100´-0˝? You are no longer working in 2D with sheets of paper; you are working in an almost infinite model space, getting ready to virtually build a structure that is more than just a line on a piece of paper.

Project Position and Orientation

It's good practice to assume that Project North is up on your display. You start out modeling your project in the orientation that best fits on your sheet. For example, if a building is skewed, you may want to orient it so that it fits parallel or perpendicular on your sheets. In this section we'll explore tools that you can use to orient your project model as needed.

Each project you model will have its own position and orientation, which are determined by its size, shape, and geographical location. Because every project will be different, this information will not be added to your templates but is a major part of setting up your project environment. If you are a structural firm or part of a MEP firm, chances are that the project position and orientation will be fully denoted by the architect. However, not every structural project is driven by an architectural concept, so you should understand what all of this means.

Since you are now working in a 3D environment, everything is tied together so you cannot just simply go to a plan view and move the entire plan over 2´ or select everything in a 3D view and move it up 1´ because the building's location needs to moved. The building will move, but all the visible portions of your sections, crop regions, 2D content, and several other items will remain in their original location.

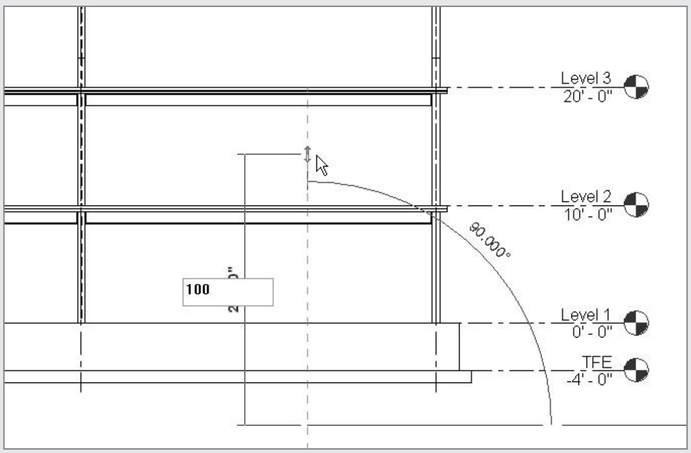
For this reason, Revit Structure provides tools that allow you to relocate, rotate, or mirror your project as needed. You access these tools by choosing Tools ➢ Project Position/Orientation.

The following exercises will take you through the basic steps needed to use a couple of these tools. For these exercises, you can use the `RST_PROJECT_POSITION.rvt` file (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`) or use one of your own projects.

Exercise: Relocate This Project

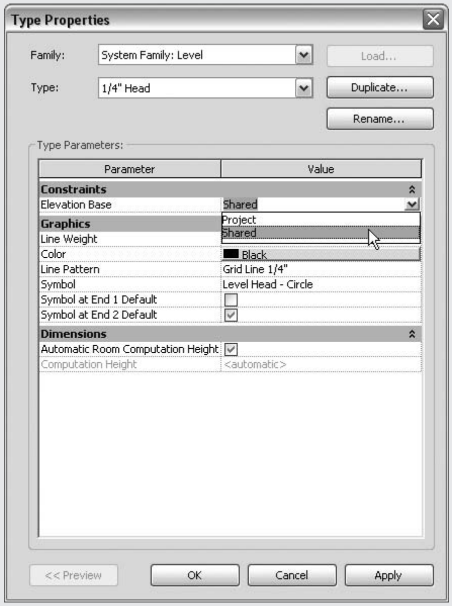
When you use the Relocate This Project tool, Revit Structure moves the entire project relative to the shared coordinate system. It behaves much like the Move command but is more powerful. No objects need to be selected or moved, and everything maintains its position and goes along for the ride. It's kind of like a moving company placing your house on a big set of wheels and moving it across town to a new location—except with Revit Structure, you can easily toggle the location of the house from its new location back to its old location using shared coordinates.

1. With the exercise file (`RST_PROJECT_POSITION.rvt`) open, go to the North elevation view. Note that a project can be relocated in either a plan or elevation view. We will be relocating the project in an elevation view.
2. Choose Tools ➢ Project Position/Orientation ➢ Relocate This Project.
3. Click anywhere in the drawing area, as shown in the following graphic, and move the cursor in an upward direction. While pulling in an upward direction, enter **100** (for 100´-0˝) into the temporary dimension and press Enter.

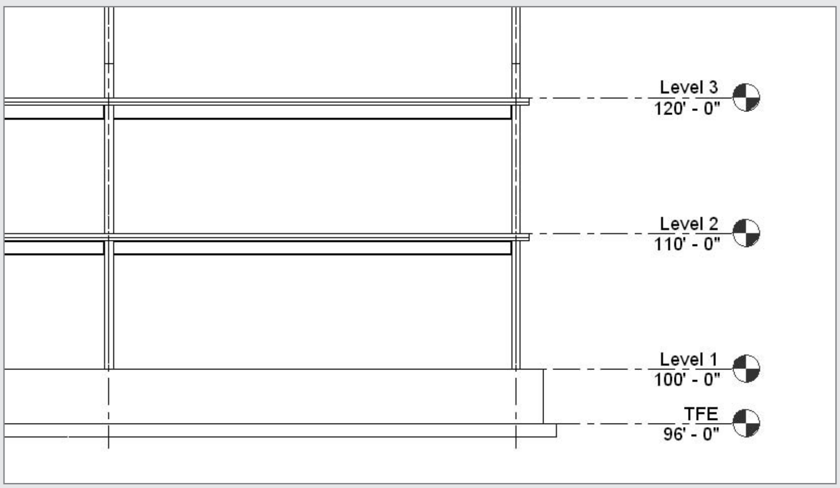


4. Step 3 just relocated your project up 100´-0˝. If the level's Elevation Base parameter is set to Project, it will appear that nothing has happened. This is because the Relocate This Project tool has set up a shared elevation and the level is currently not set to display the Shared Coordinate value.

5. Open the level's Type Properties dialog box and select the Edit/New tab. Under the Constraints group, you'll see the Elevation Base parameter. Change this value to Shared and click OK in all remaining dialog boxes.



6. All elevation references should now reflect the new Shared value that you just set. The following graphic shows that the relocation of your project is complete. At first glance, it's difficult to tell what the Elevation Base parameter of a level is set to; in this example, they are set to Shared.



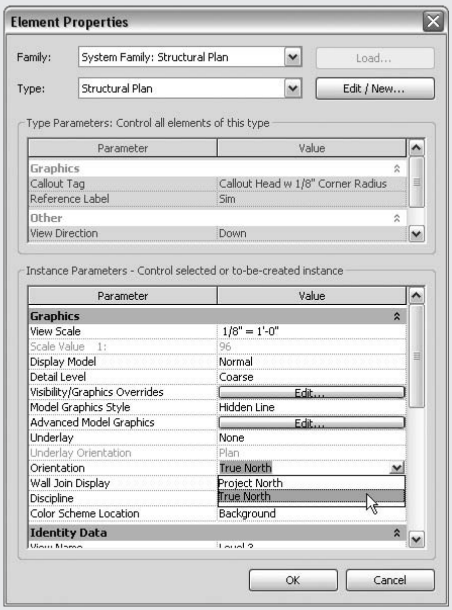
Not Everything Displays Shared

It is important to realize that you will need to change the Elevation Base setting to Project or Shared in other areas and that in some cases you cannot set Revit Structure to display the Shared Coordinate value. Levels, spot elevations, and spot coordinates will allow you to toggle the display between these two coordinate systems. The Graphical Column Schedule will not display the Shared Coordinate values; it will only display the Project Coordinates values. The Elevation at Bottom parameter value shown in the Foundation element's Element Properties dialog box will also display only the project coordinates. Take these factors into consideration before choosing this route.

Exercise: Rotate True North

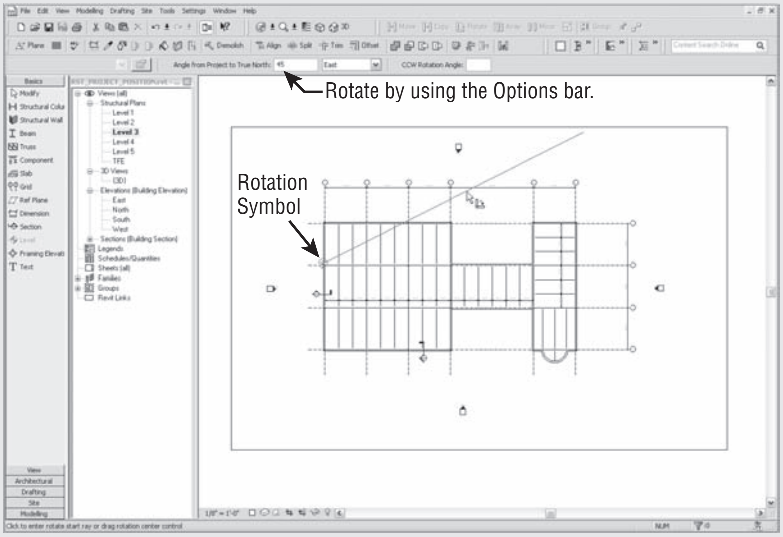
By default, Revit Structure will set all plan views to Project North. To rotate your project to True North, you will need to be in a plan view and have the view set to display True North. Rotate True North behaves much like the Rotate command, but you'll find it is more powerful.

1. With the exercise file (`RST_PROJECT_POSITION.rvt`) open, go to the Level 3 plan view.
2. Open the Element Properties dialog box of the view shown here and make sure that the Orientation parameter is set to True North.

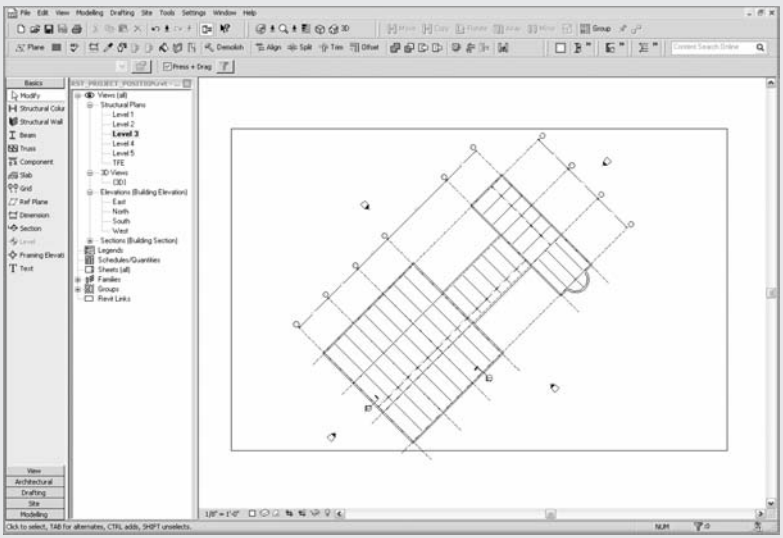


3. Select Tools ➢ Project Position/Orientation ➢ Rotate True North.

4. The default rotation symbol will display in a random location on the screen, as shown in the following graphic. You can select this symbol and move it to your chosen rotation point and rotate it as needed, or you can fill in the rotation information in the Options bar at the top of the screen. Either method will produce the same results.



5. After completing Step 4, you should see that the Level 3 view is rotated at a 45-degree angle. This is the project's True North setting. Plan views not set to the True North setting will still display at their Project North location.



6. Observe that any annotation and detail line work has autorotated to read correctly and that the other plan views are still set to Project (which means they are not rotated 45 degrees).

Mirror Project

The Mirror Project tool is something that is fairly new to Revit Structure, which means that it probably will not do 100 percent of everything you expect it to do. Exercise caution when using this tool; you may even want to try a few dry runs before running it on your live project. It behaves similar to the Mirror command. To mirror your project, select Tools ➢ Project Position/Orientation ➢ Mirror Project. You can mirror the project in several preset directions, as shown in the Mirror Project dialog box (Figure 2.14).

Figure 2.14
Mirror Project dialog box

Since this tool will mirror your project, it is going to take all annotations, section cuts, elevations, and other elements along for the ride and do its best to keep everything intact. It will not renumber grids for you so they display in the correct order or flip your section cuts on plan to maintain their original cut direction. However, it will allow you to export a list of errors that have occurred during the mirroring process so you can identify potential problem areas.

This tool is a bit different from the Relocate This Project and Rotate True North tools. It does not have a setting that allows you to flip back and forth between two different mirrored views. The tool still uses the project coordinates and preserves any shared coordinates that you may have created.

Rotate Project North

The Rotating Project North tool is similar to Mirror Project in that it too does not create shared coordinates: it rotates the project coordinates. This method will also do its best to take everything along for the ride, but more than likely you will be presented with a list of errors. These errors can also be exported so you can address them easily. Begin by choosing Tools ➢ Project Position/Orientation ➢ Rotate Project North. You can rotate the Project North in several preset directions, as shown in the Rotate Project dialog box (Figure 2.15).

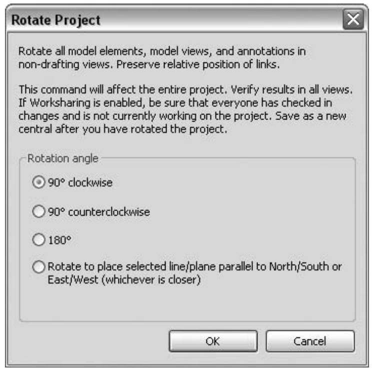


FIGURE 2.15
Rotate Project dialog box

Project Units

Depending on what area of the world you are working in, you may be using either imperial or metric units. Also depending on what types of projects you are working on, you may have set different units of precession. These units are displayed in dimensions, elevations, annotation tags, and even schedules.

Revit Structure has three areas of units that it has broken out into separate sections. You can see these sections by choosing Settings ➤ Project Settings. We will discuss two of those sections next because they pertain specifically to Revit Structure. The Electrical unit set includes such settings as Illuminance, Luminous Flux, Luminous Intensity, Efficacy, Wattage, and Color Temperature.

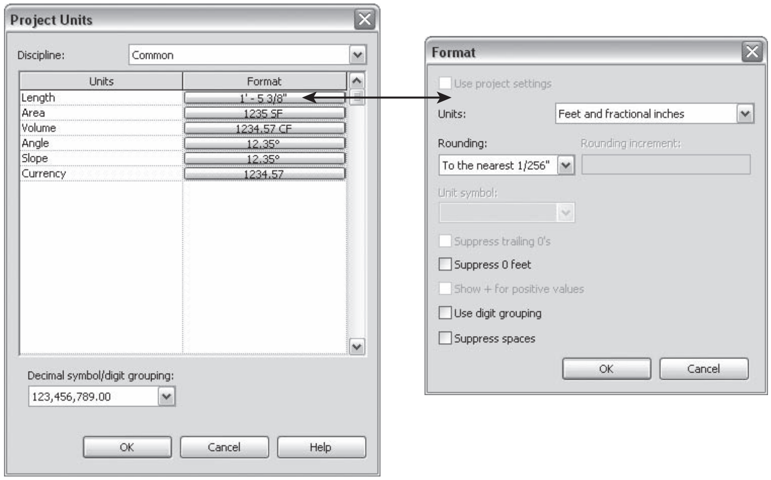
Each set of units has several settings that allow you to adapt to the region and precision that you are preparing your documents for. We will not go into great depth with all of these individual levels, but I encourage you to step through each one as well as test the behavior that each setting change results in.

Common Units

The Common units include settings such as Length, Area, Volume, Angle, Slope, and Currency. Figure 2.16 shows the Project Units dialog box with Common chosen from the Discipline drop-down. This dialog box sets the units globally for the project, but you can override them when you're listing values in schedules, tags, and dimension style types.

The Common units you see in Revit Structure are much the same as what you would see in the other Revit platforms. You will find that setting these units is similar to what you are accustomed to doing in other CAD programs. To access settings that are more specific to Revit Structure, use the Structural units, which we'll discuss next.

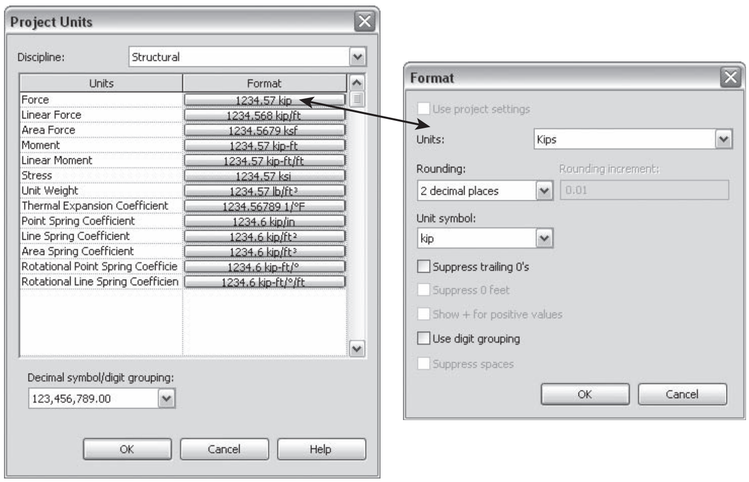
FIGURE 2.16
Setting your units for the Common discipline



Structural Units

The Structural units include such settings as Force, Moment, Stress, Unit Weight, and Coefficient. Figure 2.17 shows the Project Units dialog box with Structural chosen from the Discipline drop-down. This dialog box sets the units globally for the project, but again, you can override them when you're listing values in schedules and tags.

FIGURE 2.17
Setting your units for the Structural discipline

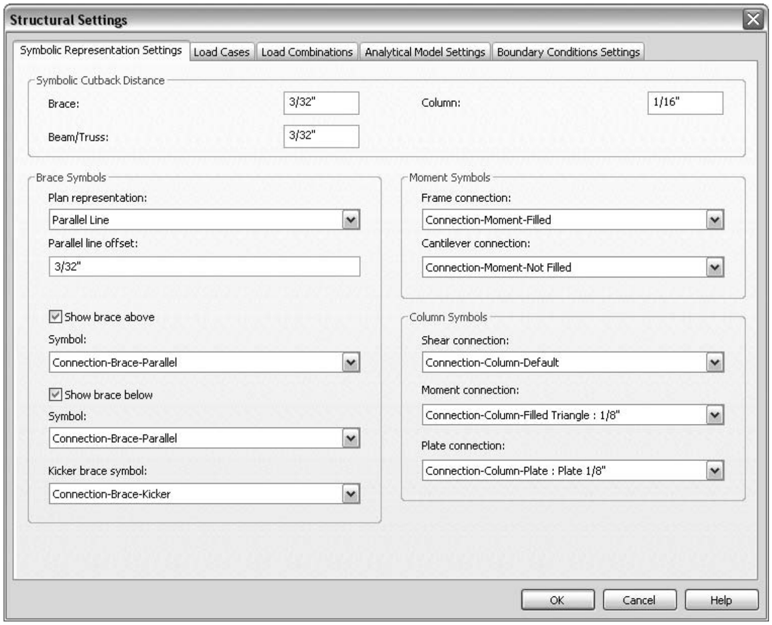


The Structural units resemble those in your analytical software. Depending on the type of work you do or where in the world your work is done, you may have separate templates that store this information to help keep your workflow efficient.

Structural Settings

The Structural Settings dialog box (Figure 2.18) contains settings for most structure-related items. You can save all information shown in this dialog box in your template files.

FIGURE 2.18
The Structural Settings dialog box



You will find that most of these settings pertain to how your documentation will look. A major aspect of showing framing for documentation involves single-line symbology as well as symbols for connection types. The next section will show how Revit Structure settings can help you adjust the symbolic representation of various structural elements for your documentation output. Following the symbolic representation settings, we will briefly discuss the settings that can be made to the analytical portions of the model.

Symbolic Representation Settings

The first tab in the Structural Settings dialog box is called Symbolic Representation Settings. Settings on this tab control the display and symbology of the different structural conditions in plans, sections, and schedules for steel shapes. These options determine how certain elements of an object display or allow you to use a symbol of your choice when certain conditions occur.

SYMBOLIC CUTBACK DISTANCE

The settings in the Symbolic Cutback Distance area shown in Figure 2.19 control the structural symbolic lines that display in a coarse detail level.

FIGURE 2.19
Setting the Symbolic Cutback Distance values

The Symbolic Cutback Distance values are global settings that affect all beams, trusses, braces, and steel columns in your project. These are all model objects with embedded symbolic line work. The Column Cutback setting will primarily be used in a Graphical Column Schedule. Figure 2.20 shows examples of what these settings affect.

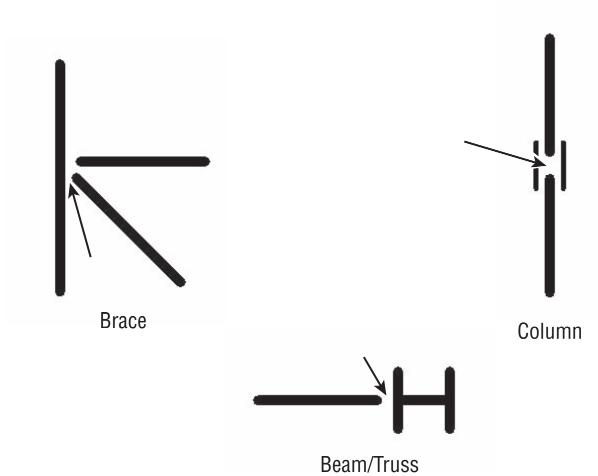


FIGURE 2.20
The Symbolic Cutback Distance values can be controlled as global settings for the project.

BRACE SYMBOLS

The Brace Symbols settings shown in Figure 2.21 control the symbolic display of bracing and kickers that exist throughout your project.

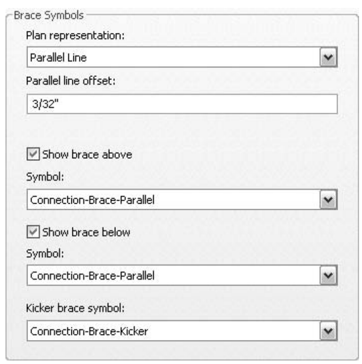
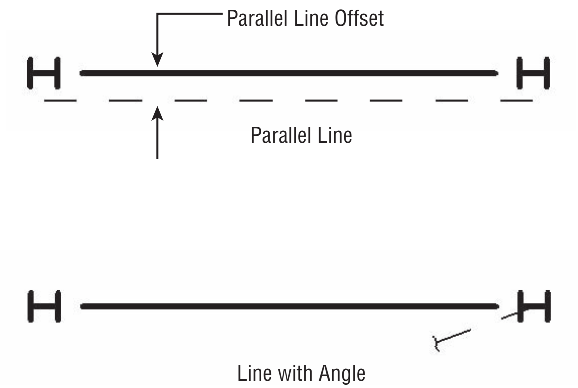


FIGURE 2.21
The Brace Symbols settings

Revit Structure automatically places a symbol for framing members in a plan view when their structural usage property is set to Vertical Bracing or Kicker Bracing but only when the Structural Framing category is set to a coarse detail level.

For vertical bracing, you can choose between two different plan representation types: Parallel Line and Line with Angle. Figure 2.22 shows an example of each. The Parallel Line option has an additional offset setting so that the symbolic line can clearly be seen on your documentation. Revit Structure displays a Line with Angle symbol for the brace above and below depending on where the brace is connected.

FIGURE 2.22
Plan representation of bracing



THAT IS NOT HOW WE SHOW OUR BRACING

If these two symbolic representations do not fit within your company's standard display of bracing, you can create a new symbolic bracing symbol by using the Generic Annotation family. If creating a new symbol still does not give you the look you want or you dislike Revit Structure's behavior in displaying these symbolic representations, then you can deselect the Show Brace Above and Show Brace Below check boxes. This will allow you to use detail lines or other methods to achieve the look you desire.

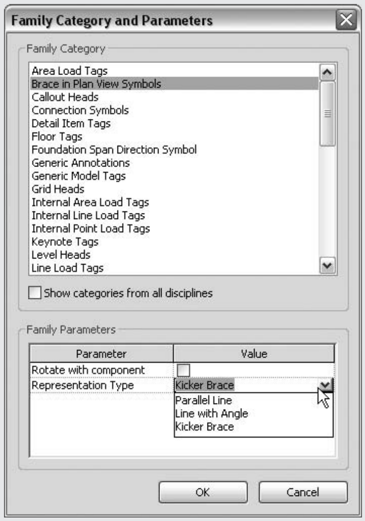
The Kicker Brace symbol functions a bit differently than the other bracing symbols. Setting the plan representation does not have any effect on it, and you cannot toggle its above and below display. A kicker brace displays a symbol of your choice at the end of the kicker or at the kicker's uppermost connection point. Figure 2.23 shows Revit Structure's default Kicker Brace symbol: an "X."

FIGURE 2.23
Plan representation of a kicker brace

Exercise: Create a New Plan View Symbol for a Kicker Brace

If you choose to have a different look for the symbolic display of a kicker brace you can do so by creating a new Brace in the Plan View Symbols family.

1. Select File ➢ New ➢ Family.
2. Browse to the `Generic Annotations.rft` family template located in the Imperial Templates - Annotations folder.
3. Choose Settings ➢ Families Category and Parameters.
4. In the Family Category and Parameters dialog box shown here, select Brace in Plan View Symbols for the category type. This sets the category type of the family so it will show up as a symbolic symbol in the Structural Settings dialog box under Brace Symbols.



5. From the Representation Type pull-down under Family Parameters, select Kicker Brace. If this option is set to anything else, this representation type will not be available in the Kicker Brace Symbol pull-down.
6. Add line work as needed for your symbolic display and delete the block of note text that was pre-loaded in the template.
7. Save the family and load it into your project.

Note that the same steps apply for creating a new Brace symbol except that in Step 5 you will need to select a Representation Type setting of Parallel Line or Line with Angle.

Moment Symbols

The settings in the Moment Symbols area shown in Figure 2.24 control the symbolic display of moment connections for framing that exist throughout your project.

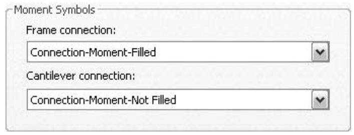


Figure 2.24
The Moment Symbols settings

These symbols only become visible when the detail level of the Framing category is set to Coarse and the properties of the element are set such that the connections are applied. Figure 2.25 shows where these connections can be assigned for each instance of a framing member.

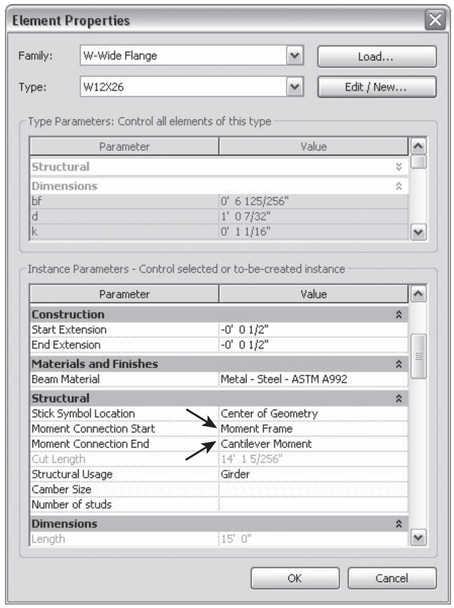


Figure 2.25
Assigning the symbolic display of a moment connection to a framing member

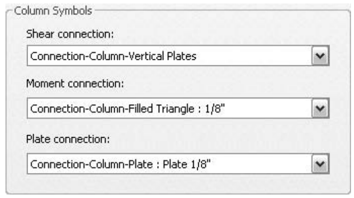
A Moment symbol falls under the category of Connection Symbol. Revit Structure has two types of framing member moment connections available: Moment Frame and Cantilever Moment. You are able to create any symbol you want for each of these two conditions. By default, Revit Structure has one set to a filled triangle and the other set to an open triangle (Figure 2.26).

Figure 2.26
Moment connection symbols in plan view

Column Symbols

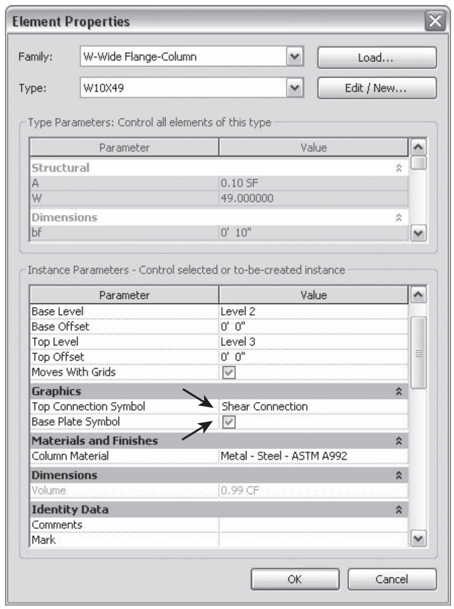
The settings in Column Symbols area shown in Figure 2.27 control the symbolic display of connections for columns that exist throughout your project.

Figure 2.27
Setting the display of Column symbols



These symbols only become visible when the detail level of the Framing category is set to Coarse and the properties of the element are set such that the connections are applied. Figure 2.28 shows where these connections can be assigned for each instance of a column member.

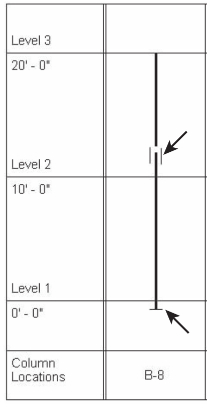
Figure 2.28
Assigning the symbolic display of a connection to a column member



A Column symbol falls under the category of Connection Symbol. Revit Structure has three types of column member connections available: Shear Column Connection, Moment Column Connection, and Base Plate Symbol. You are able to create any symbol you want for each of these three conditions.

You can make as many different Top and Base connection types as you want, but you will only be able to display a maximum of two Top Connection symbols and a maximum of 1 Base Plate Connection symbol per project. Figure 2.29 shows examples of Revit Structure's default column symbols.

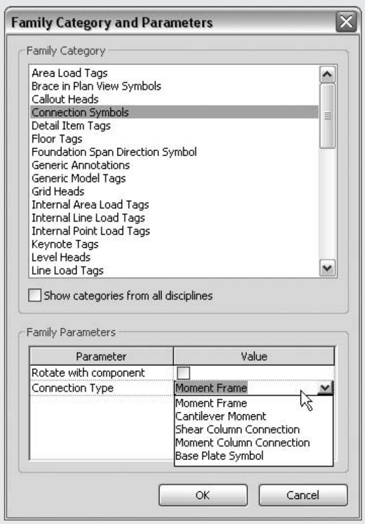
Figure 2.29
Symbolic Column symbols in Graphical Column Schedule



Exercise: Create a New Plan View Symbol for a Moment Connection

If you choose to have a different look for the symbolic display for moment connections you can do so by creating a new Connection Symbols family.

1. Choose File ➢ New ➢ Family.
2. Browse to the `Generic Annotations.rft` family template located in the Imperial Templates - Annotations folder.
3. Select Settings ➢ Families Category and Parameters.
4. In the Family Category and Parameters dialog box shown here, select Connection Symbols for the category type. This sets the category type of the family so it will show up as a symbolic symbol in the Structural Settings dialog box under Beam and Column Moment Symbols.



5. From the Connection Type pull-down under Family Parameters, select the type of Moment symbol you will be creating. Choosing Moment Frame or Cantilever Moment will classify the symbol as a Brace symbol. Choosing Shear Column Connection or Moment Column Connection will classify the symbol as a Column symbol, which will only apply to the top of a column. Choosing Base Plate Symbol will classify it as a Column symbol, which will apply to the base of the column.
6. Add line work as needed for your symbolic display and delete the block of note text that was preloaded in the template.
7. Save the family and load it into your project.

Analytical Settings

The remaining four tabs in the Structural Settings dialog box deal with the analytical portions of your project. If your project will not be taking advantage of the analytical capabilities Revit Structure offers, these settings will not matter much to you.

However, if you will be using Revit Structure for its analytical capabilities and will be placing loads inside the Revit Structure model, you will want to become familiar with these four tabs: Load Cases, Load Combinations, Analytical Model Settings, and Boundary Conditions Settings. The settings on each tab are specific to its heading.

To learn more about these settings and how to add load cases and load combinations, go to the help sections in Revit Structure and search for "Structural Settings Dialog." The help sections serve as an excellent resource for learning about these settings so that you can incorporate them into your project templates.

Rebar Settings

Revit Structure has a separate area for making settings for modeling and tagging of reinforcing bars (rebar). You'll learn more about the use of these settings and tools in Chapter 10.

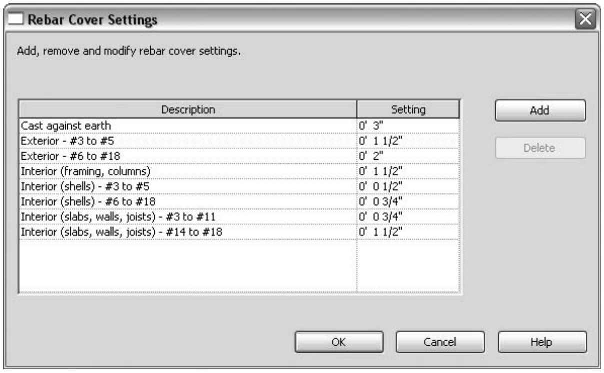
Revit Structure helps automate the placement of rebars as well as how a rebar is annotated when tagged. Part of this automation is allowing the user to set rebar coverage types that can be applied to different concrete types and their conditions. This helps keep 3D rebar within the concrete element as shapes change in size and location. Once a rebar is placed, it can be tagged with predefined standard abbreviation callouts based on its placement properties.

Adjusting the Cover Distance

To set up the rebar cover distance for your project, select Settings ➢ Rebar Cover Distance to open the dialog box shown in Figure 2.30.

By clicking the Add button, you are able to make as many different cover types as you want, depending on the various conditions that may occur for your project. You need only specify two settings: you must enter a description and the dimension of cover from an outside edge of a concrete member to the face of a rebar element. A well-thought-out naming convention will go a long way in helping users quickly set the proper cover required throughout the project.

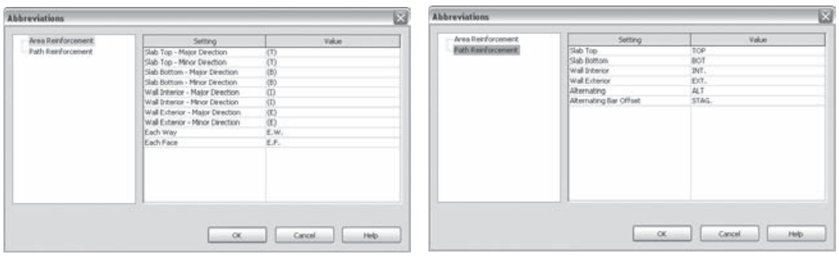
FIGURE 2.30
Setting up the rebar cover for your project



Abbreviation for Tagging

When it comes to annotating the rebar that you have modeled, Revit Structure has a few built-in abbreviations that tags will automatically pull from depending on the properties of the rebar. To get to these settings, choose Settings ➢ Abbreviations. A dialog box opens whose settings reflect whether the rebar uses area or path reinforcing. Figure 2.31 shows the options for both area and path reinforcement.

FIGURE 2.31
Setting abbreviations for reinforcing tagging



You are not allowed to create new settings, but you can revise the value of the ones Revit Structure provides to match your company's standards. Making use of these settings and annotating with them helps keep your documents looking consistent no matter who is producing them.

Options

Settings in the Options dialog box are specific to the machine that Revit Structure is installed on. These settings cannot be incorporated into your template files. They deal more with how the program functions and how you interact with it rather than project-specific settings. The majority of these settings are stored in the `Revit.ini` file, which is usually located in `C:\Program Files\Revit Structure 2009\Program\`. We'll cover this file and its uses in Chapter 17.

To open the Options dialog box (Figure 2.32), select Settings ➢ Options. You'll see several tabs across the top, which allow you to set own working environment settings (they will not affect other users working on the project). On the General tab, you will be able to set up save reminders and specify your username for workshared models, among other settings.

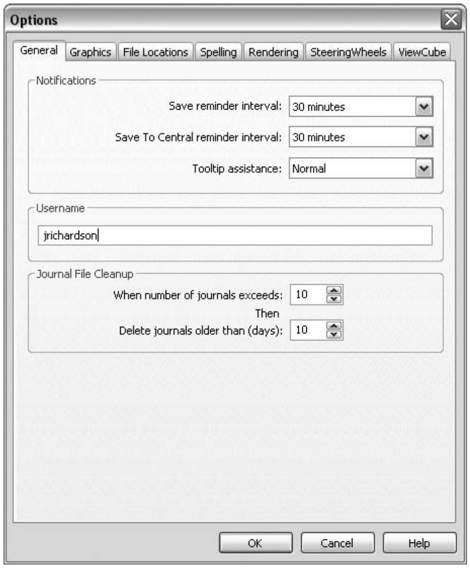


FIGURE 2.32
User-specific settings on the General tab of the Options dialog box

If you still want to work with a black background, click the Graphics tab (Figure 2.33) and select the Invert Background Color check box. If you don't like the color of objects when selected or the object warning color when objects overlap, you can also change those settings on this tab.

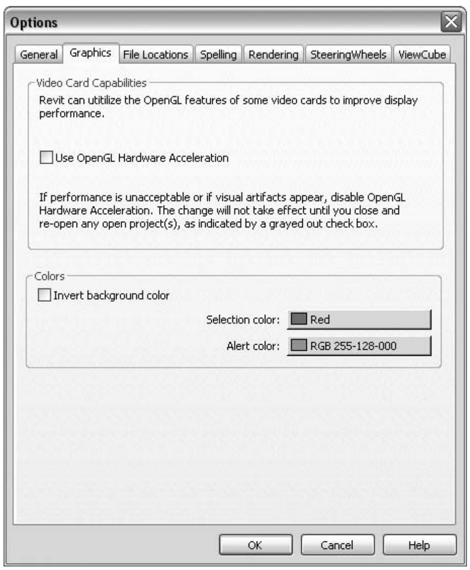
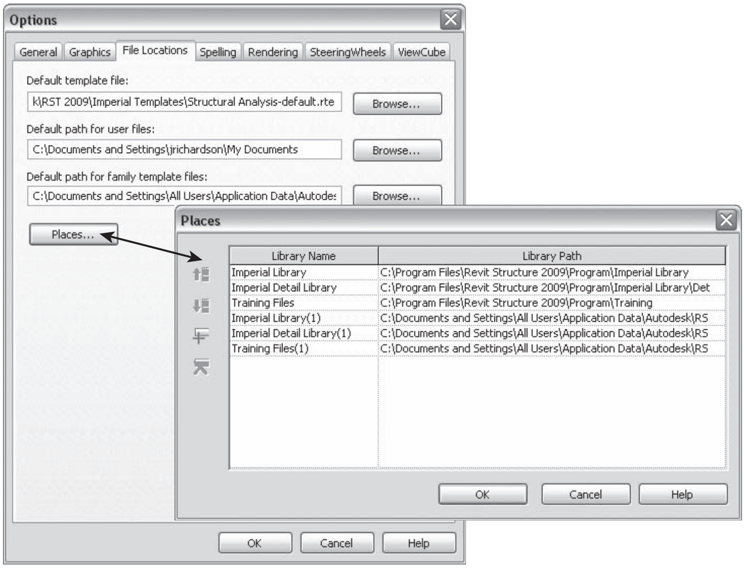


FIGURE 2.33
User-specific settings on the Graphics tab of the Options dialog box

The File Locations tab (Figure 2.34) lets you set default locations for certain files so Revit Structure can quickly take you there when you use any Load command or start a new project. You can also click the Places button to set up your own shortcuts to folder locations; these shortcuts will appear in the Open and Load dialog boxes.

FIGURE 2.34
User-specific settings on the File Locations tab of the Options dialog box



Become familiar with the settings on all the Option dialog box tabs so you can configure Revit Structure to function independently for each individual user. Also, use the Revit Structure Help documentation to further your knowledge regarding these settings.

Project Browser Organization

It doesn't take long to figure out that, once you are working in Revit Structure, the Project Browser needs some type of organization or things will get awfully messy, very fast. The first thing you should do is devise a naming convention for your view names; second, use folders and filters to help display your views in the Project Browser in a way you can quickly and easily access them.

View Naming Conventions

Firms using Revit Structure have come up with naming conventions specific to their own working methods as well as project type. A good naming convention should be developed in every Revit Structure environment, but it will have to be revisited for each project.

Revit Structure does its best to automatically name views specific to their view type. For example, a new plan is named after the level, a new detail is called Detail 1 or Detail 2 (depending on which number is next), a new section follows the same rules (except it is Section 1), a new elevation is called Elevation 1-a or Elevation 1-b, and a drafting view is called Drafting 1 or Drafting 2. Figure 2.35 shows Revit Structure's way of helping you develop a naming convention.

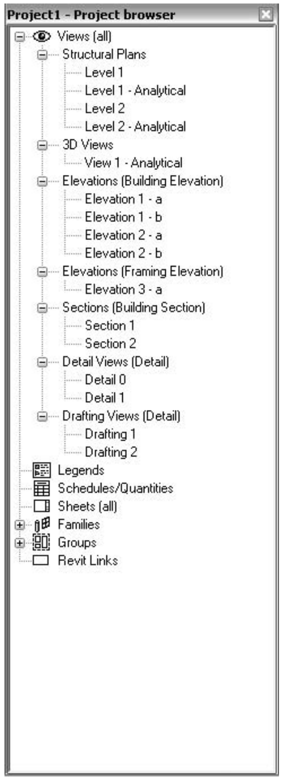


FIGURE 2.35
Revit Structure default naming conventions

This helps, but eventually the numbers can get out of sequence and make it difficult to find anything. As views are created and take on different roles, you will want assign them names that reflect those roles. Here are some examples:

Working_Detail 1 A detail used for working or looking at the model

Ref_Detail 1 A detail used to help lay out the model

Coord_Level 2 A plan used for coordinating with other disciplines

Export_Level 1 A plan used for exporting to another format

CD_Level 1 A plan used for construction documentation

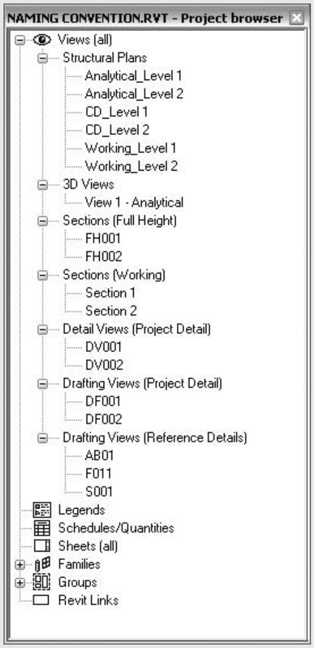
ACAD_Level 2 A plan where DWG files are linked into

No naming convention will be handed to you on a sheet of paper that says, "This is how you name your views"; it is important for you to know this and for you to make the effort up front to develop one that works best for you.

Real World Scenario

One Approach to a Naming Convention

We have taken an approach that pretty much overrides Revit Structure's default naming convention. When a user creates a new view, the first thing he or she does is assign that view a name specific to the role it will be taking on. The use of this method along with some basic browser organization helps keep our projects neat and clean to work with.



We have created various section, detail, and drafting view types to help us sort them in the browser. If it's a detail, the view is given the type Project Detail and the name DV### (for Detail View). Drafting views follow the same method, only with a DF###. The use of Revit Structure has allowed us to display more Full Height sections, so a different section type has been created for them as well (and they take on the name FH###). The Working type sections are meant to be temporary and should never be put on sheets, so they take on whichever name Revit Structure gives them.

As we get further along in our implementation, we may revise and adapt our naming convention to reflect the exciting changes Revit brings and our new working methods.

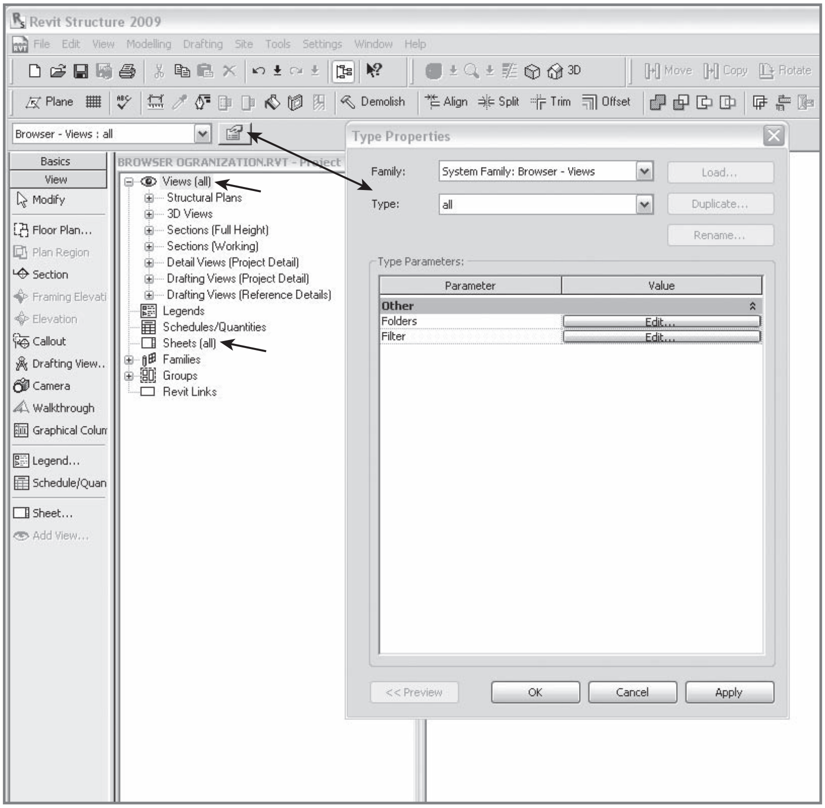
Common Methods of Organization

Chapter 1 briefly touched on creating a new browser view type to show you the views that have been placed on a sheet. You switch to this view type when you are only going to be working with these types of views. This method filters out all those unwanted views so you do not need to weed through them. Using the Project Browser in this way will help increase your productivity as well as allow you and others to move efficiently between views.

In this section, we will take that method one step further and show you how to add your own parameter to a view and/or drawing sheet to allow you to sort your views in ways that can be more specific to your task at hand.

Just like every other object or element in Revit Structure, the Project Browser also has its own type. One way to get to the browser type list is to choose Settings ➢ Browser Organization. Another way, as shown in Figure 2.36, is to select the View or the Sheets category in the Project Browser and open the properties for it.

FIGURE 2.36
Accessing the browser view types



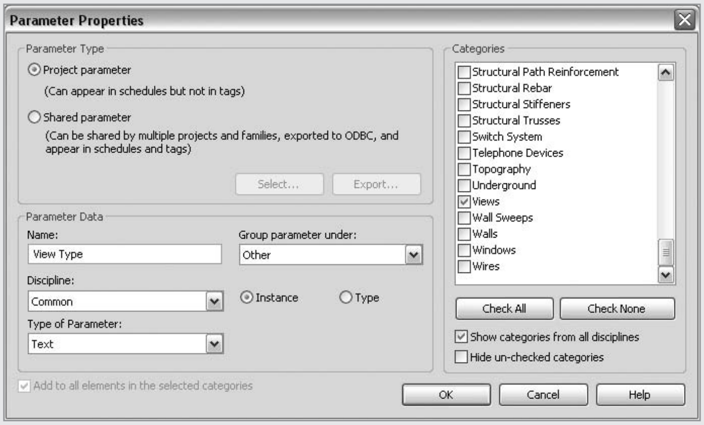
Different view types will have to be set up depending on whether you will be sorting views or sheets. These are the only types that you are allowed to sort. You cannot sort legends and schedules because they have different view properties assigned to them.

We will walk you through a couple of quick exercises designed to get you thinking about all of the possibilities you could incorporate into your working environment.

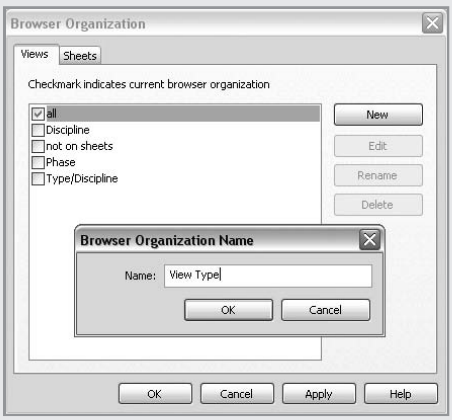
Exercise: Creating a New Browser View Type

For this project view type, we are going to create a new project parameter called View Type, which will be applied to the category View. We will then set up a new Project Browser view type to help you sort your views in the browser.

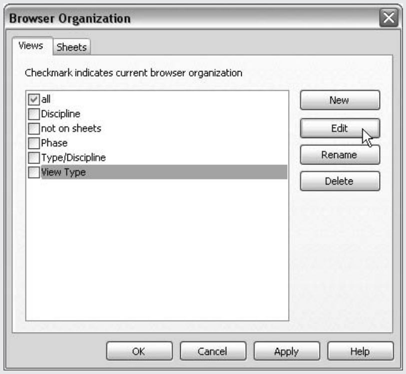
1. While in your project, choose Settings ➤ Project Parameters.
2. Click the Add button and create a new project parameter called View Type with the properties shown in the following graphic. Click OK to close this dialog box and then again to close the Project Parameters dialog box.



3. Choose Settings ➤ Browser Organization and click the New button on the Views tab.
4. In the Browser Organization Name box, enter **View Type** and click OK. Be sure to assign names that reflect the organization methods.



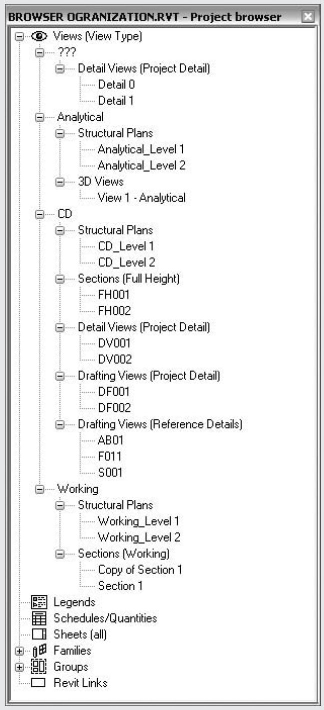
5. Select the new browser organization name as shown here and click Edit.



6. On the Folders tab, fill in the fields as shown in the following graphic. In the properties of a browser organization, you can set up folders to group and sort views by.

7. In this case we will not use the Filter tab, but you could apply filters to the browser view type so certain views will be hidden from view. The following dialog box shows a filter applied that would only display views that have a value of CD in the View Type parameter.

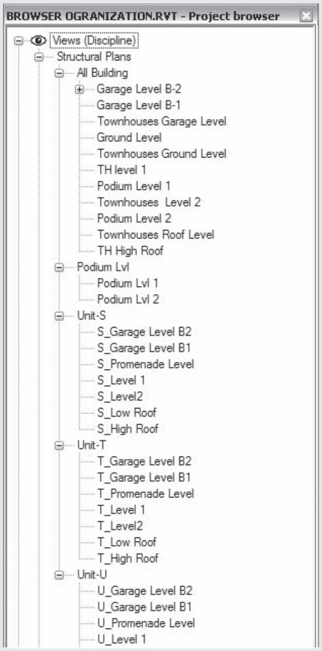
8. Make your new browser view type the active type.
9. Open the properties option of your views and put in a value for the View Type parameter. Watch the Project Browser reorganize into folders as you add these values. Parameters that are left without values will automatically be grouped under a "???" folder designation. The following graphic shows the browser window sorting by View Type.

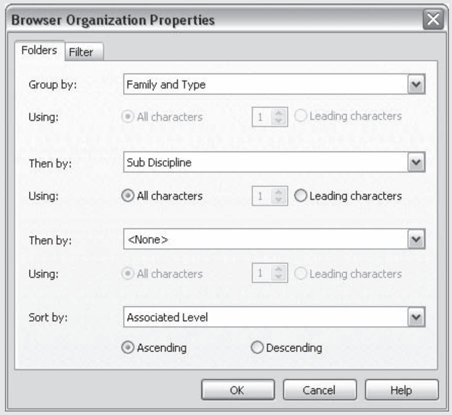


To create a browser view type for sheets, you will follow the same procedure shown here except you will apply the project parameter to the Drawing Sheets category instead of Views and use the Sheets tab in the Browser Organization dialog box.

Another Approach to Browser Organization

This project consists of 25 stories of high-rise condos, with three levels of townhouses at the lower levels. Everything is sitting on a big podium over parking levels.





In this example we sort the browser by grouping the family and type name than by a Sub Discipline parameter that is added as a project parameter to the views. The value of the Sub Parameter would be Podium Lvl, Unit-S, Unit-T, and so forth. In the groupings we sort by the view's associated level in ascending order. This organizes the plans from bottom to top in the browser.

Transferring Project Standards

Transferring project standards is another way of getting data from a template file or another project into your current project without having to re-create the setup. You can use the same method to transfer current standard settings that may have changed in projects with old standards to bring them up to date.

To do this, you need to have the file you will be transferring data from open at the same time as the project you will be transferring data into. Keep in mind that if both projects that are open are large in file size, you might not want to do this—you could bring your computer to a screeching

halt. To get around this, start a new project from a None template and transfer the settings into that. Save this new file and then use it to transfer the data into your other project. The new project that was created from the None template will be much smaller in size and will allow you to clean anything out of it you do not want transferred.

To transfer projects standards, choose File ➤ Transfer Project Standards while you are in the project that you want to transfer standards into. When the dialog box shown in Figure 2.37 opens, click the Copy From box and select from a list of templates or projects that are currently open. In the panel below, only check the categories that you want to transfer. In this case, we will transfer from the Concrete Wall Types template and we will transfer only wall types.

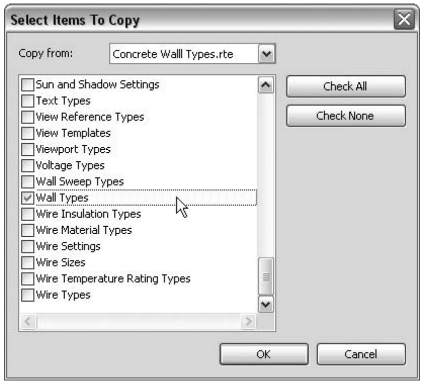


FIGURE 2.37
Transferring standards from the Concrete Wall Types template file

As shown in Figure 2.38, if any standards currently exist in your project, you will be given the option to only add the new ones or to overwrite the ones that are already there.

FIGURE 2.38
The Duplicate Types options when you transfer project standards

Transferring settings and already customized families from one project or template to another is a quick way to keep standards applied throughout your projects. It will also eliminate redundant creation of system family types like walls, slabs, and roofs, and annotation types such as text, dimensions, and spot dimensions. Taking advantage of this feature will help you keep each project up to date with your latest project environment settings.

The Bottom Line

Develop your own custom templates Knowing all of the items that you can store in a template file will help you avoid creating data that is the same over and over and force company standards.

Master It What extension does a template file have? What types of things can be stored in a template file? What cannot be stored in a template?

Set project units and precision display Setting the units for your project and the precision of them is part of documenting your model for those who will be using it.

Master It What are the three types of units in Revit Structure?

Make adjustments to structure-specific settings Revit Structure has several areas of settings that are specific to how elements display for your documentation.

Master It Where do you go to assign the global display of symbolic symbols for different types of connections? What type of family template do you use to create a new symbol type?

Organize the Project Browser Learning how to organize your browser depending on the workflow and requirements of the project will allow you to work more efficiently and keeps the browser free and clean of unused views.

Master It For what two things can you create a browser view type in the Project Browser? When creating a new project parameter so that you can sort your sheets in the Project Browser, what category do you need to apply the new parameter to?

Transfer standards into your project Being able to use settings and content from past or other current projects allows you to avoid duplicating your efforts over and over again and ensures that standards are kept.

Master It What command is used to bring new line weight settings into a project that is using old line weight settings?

Chapter 3

Starting to Model Your Project

Planning? Yes! Before you begin to develop your Revit Structure model, you should understand the world in which this modeling will occur. In this chapter you will examine the basic functions and features of Revit Structure that enable you to plan your model the proper way—the first time.

The fundamentals of Revit Structure will set the tone of your entire model and for everyone who must interact with it. A misstep here and your model, at best, will annoy others and, at worst, make completing your work in a timely fashion impossible. Imagine a contractor building a structure from the top down; it would make for a very expensive proposition. Revit Structure also provides methods that let you perform wonders.

In this chapter you will learn to:

- Import and link CAD data
- Link Revit Architecture
- Create structural grids
- Create levels

Importing and Linking

For decades, engineers have utilized computer-aided drafting and design programs to develop plans, sections, and details for construction drawings. They have been used to design a backyard shed, all the way up to the very largest buildings in the world. It is this plethora of data types that can gain new life—as a backbone for generating BIM data within Revit Structure. If you previously created AutoCAD drawings of a project that now needs a new addition, you can leverage that data by importing it. And if you have drawings from an architect who has not made the transition to Revit Structure, you can still leverage that 2D line work to build new, smart Revit Structure objects.

Data Formats

Software programs in today's world come with their own specific file formats. Each has its own respective "code" that enables the originating program to read and display the data contained within it. Revit Structure can understand several formats that can be imported for use or display purposes. These various file types contain vector data:

DWG The native format for Autodesk AutoCAD and AutoCAD LT, DWG has been the de facto standard in the building industry to date. Once the drawing has been imported or linked, you can reference objects within the single file element or even explode it into subelements.

DXF The Drawing Exchange Format was also created by Autodesk but is used more as a shared file format by software developers. Because DXF is a more widely available format, you will often find other programs, such as Computers & Structures' engineering analysis program SAP2000, that can export directly to `.dxf` and then into Revit Structure.

DGN The native format for Bentley Systems' MicroStation, this vector file format has been used for thousands of larger commercial, infrastructure, and industrial projects.

SAT The premier format for solid modeling and machining, the ACIS specification is by Spatial. Most often used for the aerospace industry, the SAT format has been given new life by major design firms that specialize in nonlinear surface design.

SKP A new format to the world of Computer Aided Design, Google's SketchUp is a free facet modeler often used by architects because of its ease of use and viewing tools. It too can be imported into Revit Structure but generally only for volumetric objects contained within an in-place family.

To import any of these file types, select File ➤ Import/Link ➤ CAD Formats. The Import/Link CAD Formats dialog box opens (see Figure 3.1), where you can define how best to use your data file.

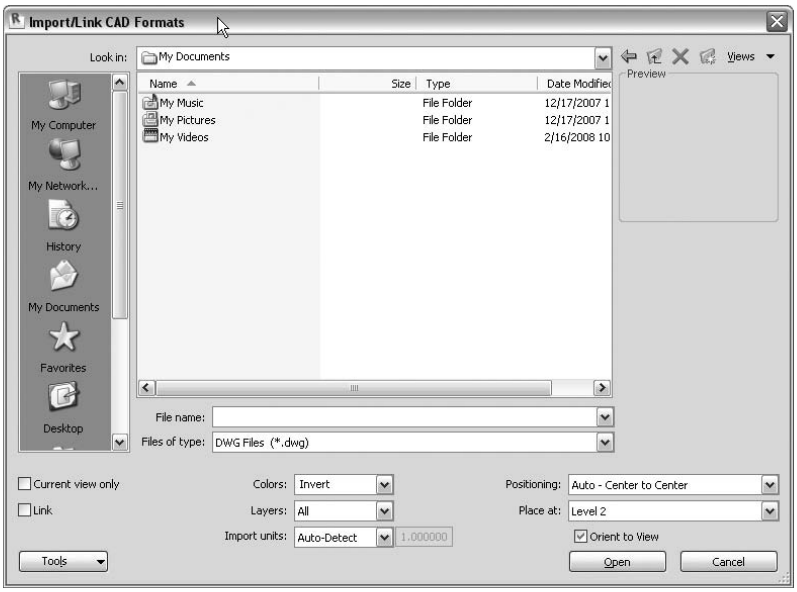


FIGURE 3.1
The Import/Link CAD Formats dialog box enables you to import various forms of vector data for use in Revit Structure.

IMPORTING DWG DATA FOR ARCHITECTURAL BACKGROUNDS

If you are already using Revit Structure but the architectural firms you work with are not using Revit Architecture, this does not leave you with a dilemma. You can expect that 90 percent of your projects will start with an architectural background that originated either in AutoCAD or Revit Architecture.

When you're inserting or linking a DWG file, you must consider several factors before executing the import. One is the level in which the DWG file will reside. In Revit Structure, each level

constitutes a floor plan. It makes sense then, if the first floor of an architectural plan is to be inserted into Revit Structure, the current floor should be Level 1. Creating levels will be discussed later in this chapter. By default, Revit Structure will start with Level 1 and then Level 2, as illustrated in Figure 3.2. For this example, Level 1 will be used to import the CAD background.

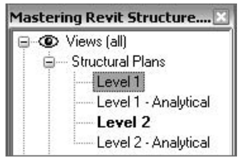


Figure 3.2
By default, Revit Structure begins with two levels. It is important to know which level you are importing the DWG into.

Another factor you need to consider is what is the DWG will be used for. You can either import or link the DWG. When you use the import function, you are basically inserting the DWG as what is referred to in AutoCAD as a *block*. This means that when the DWG file is brought into the model, it maintains no link to the original DWG. If you link the DWG, it will maintain a live path to the original DWG. If you choose to import the file, chances are you are using the DWG as a detail or something other than an actual floor plan. If you choose to link the DWG, this means that you more than likely are using it as an architectural background. If the DWG file is updated, you can reload it in Revit Structure. This behavior is almost the same as with the AutoCAD external reference method. This translates to one less process you need to change in order to utilize BIM.

Once you have decided where the DWG is going to be placed, and what it is going to be used for, the rest is a straightforward process. If your choice is to import the DWG as an architectural underlay, select File ➢ Import/Link ➢ CAD Formats, as Figure 3.3 shows.

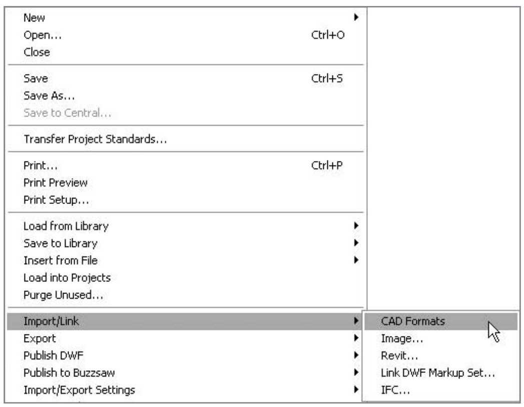


Figure 3.3
To add a DWG to the model, select this command.

An old habit is to simply browse for the file, then double-click it. In AutoCAD, the important dialog box appears after the Browse For dialog box. In Revit Structure, all of the insert choices occur in the Import/Link CAD Formats dialog box. Be careful not to just double-click on the file once you find it.

At the bottom of the Import/Link CAD Formats dialog box, you will see a few choices, as shown in Figure 3.4.

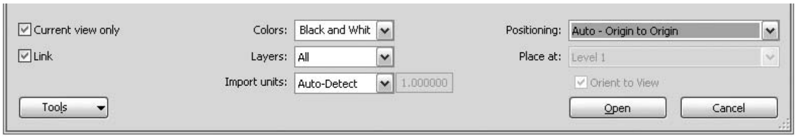


FIGURE 3.4 As you are importing the DWG, pause at the Import/Link CAD Formats dialog box to make the appropriate choices.

Consider each of these choices carefully—they are mostly irreversible once the DWG is in the model. The choices, as listed in order of appearance from right to left, are as follows:

Current View Only This will keep the DWG file specific to this view only. Choosing this option is especially important if you are going to insert multiple floor plans on different levels.

Link As mentioned, linking the DWG will keep a live path back to the original DWG. Because this is an architectural underlay, chances are that you are going to want to link this file.

Colors The choices for colors are:

Invert This option will take the original colors and find the opposites of each color. This is usually done if there are two separate underlays in the same view. You will be able to differentiate between the two.

Preserve This option will keep the original colors the same.

Black and White This option will convert all the colors to black and white.

Layers What's that? Layers? Yes! Revit Structure handles layers quite well. Normally you can select All here and deal with the layers from within Revit Structure. This feature will be explored in a moment.

Import Units When you are importing a DWG, you are dealing with CAD. This means scale becomes a nuisance again. If you select Inches, 9 times out of 10 you will achieve the correct scale. Always measure a distance after you get into Revit Structure anyway, just to be sure. If you are bringing in a site plan or topography, you might have to use feet.

Positioning Positioning basically deals with coordinates. Even in Revit Structure, there is still a need to deal with coordinates. There are several choices upon import:

Auto – Center to Center This option will simply find the bounding box (crop region) of that view, and center the DWG based on the calculated center of the view.

Auto – Origin to Origin Revit Structure finds the import's world origin, and places it at the project's internal origin.

Auto – By Shared Coordinates Revit Structure will place the imported instance based on coordinates that have been shared between the Revit Structure model and the import being brought in. Normally, the first imported instance will not have its coordinates shared with the Revit Structure model. If you have not shared the coordinates at this point, Revit Structure will inform you that no coordinates have been shared and that the DWG will be placed using the drawing's world coordinate system (Auto – Origin to Origin).

Manual – Origin The inserted DWG is placed basically at the DWG's 0,0 point. You then manually pick a point in the Revit Structure model to place the DWG.

Manual – Base Point The DWG's base point is centered on the cursor. This is common if the import is a block. You then manually pick a point in the Revit Structure model to place the DWG.

Manual – Center The inserted DWG is centered around the cursor.

Note that, as you are inserting a DWG instance, there is a good chance that once you place the DWG, it may not be visible to you. The crop region of the view may be cropping out the placement of the AutoCAD drawing. Simply turn on the crop region and stretch it to find the import, or turn it off completely and show everything within that view.

Place At If you did not select the current View Only check box, you can select a level to place the DWG on, different from the level you may currently have open.

Orient to View If the orientation of your view has been rotated, Revit Structure will rotate the DWG based on the rotation of the view. This is most common for a site or topography.

Once the DWG has been added to the model, it can be better coordinated. One of the best methods of coordinating a placement of a linked DWG is to share the location of the drawing file after it has been imported.

Assigning Placement

By specifying a shared location, you can ensure that, if the drawing data has been moved either in the Revit Structure model or in the AutoCAD drawing file, you can still maintain a specific location in the model. A good example is when a DWG is placed in the model, then moved to a correct position. Once the position has been established, you can save this location internally within the model. The next time you insert a DWG that should match the location of the existing DWG, you can select Auto – By Shared Coordinates as you link the DWG (see Figure 3.5).

Figure 3.5 Once you have set up a shared coordinate system, the next time you bring in a DWG, you can select Auto – By Shared Coordinates.

To set up a shared location, select the DWG import, then click the Element Properties button on the Options bar. There are not many parameters you can change with a linked DWG, but one of the parameters you have access to is Shared Location. To the right of the Shared Location parameter is a button. By default, it will say Not Shared, as Figure 3.6 shows.

Once you click the Not Shared button, you will see a dialog box that contains two choices. The first choice is Publish, and the second is Acquire.

When you select Publish, you are choosing to save the current location. Revit Structure will send this location to the AutoCAD drawing file as well. If you click Publish, it is a good idea to click the Change button under Record Selected Instance as Being Positioned at Location, as shown in Figure 3.7.

FIGURE 3.6
The Element Properties dialog box contains a Shared Parameter button, which allows you to record the location of the DWG.

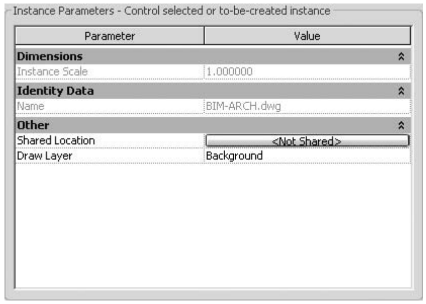
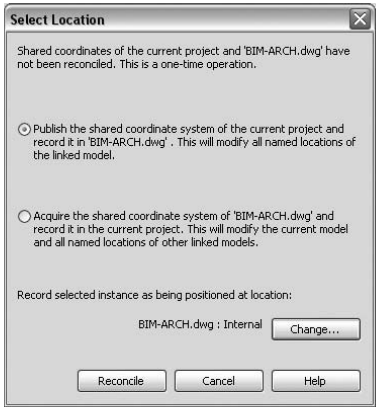
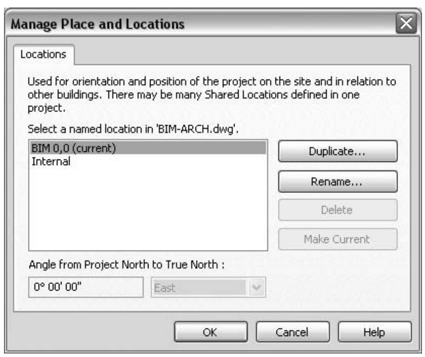


FIGURE 3.7
The Select Location dialog box will either save a new location or allow you to change the location of an inserted DWG based on an existing shared location.



The next dialog box is Manage Place and Locations, shown in Figure 3.8. You can access the same dialog box by choosing Settings ➢ Manage Place and Locations. The default location is called Internal. If you click Duplicate, you can specify a new location. Once you've created the new location, click Set Current, and you have saved the location.

FIGURE 3.8
You can access the Manage Place and Locations dialog box through the element properties of the DWG, or you can choose Settings ➢ Manage Place and Locations.



Once you are done managing the place and locations, click OK, then click Reconcile in the next dialog box. This will record the location of the DWG. At this point, when you save the actual Revit Structure model (we recommend you do so as soon as possible), you get a dialog box asking if you want to record the saved location in the DWG file, as shown in Figure 3.9.

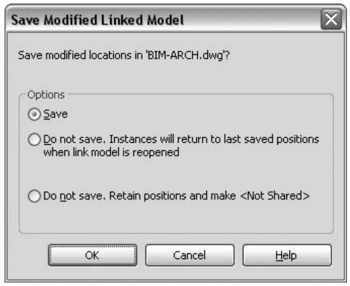


FIGURE 3.9
When you save for the first time after adding a shared location, Revit Structure will prompt you with the Save Modified Linked Model dialog box.

With the location created, and the Revit Structure model saved, you can unpin the DWG insert and physically move it. If you select the DWG, open the Element Properties dialog box, and click the Shared Locations button, you will see a dialog box asking you if you want to relocate the shared coordinate to the new location, or if you want to move the DWG back to the named location, as shown in Figure 3.10. If you click Move Instance To and then choose the named location, the DWG will be moved back to the original spot.

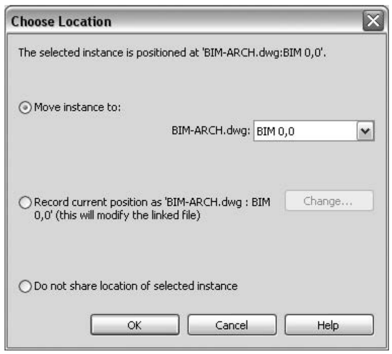


FIGURE 3.10
If the DWG moves, you can access the Choose Location dialog box through the element properties.

Now, when you link another DWG, you can use the Auto – Shared Coordinates option, and Revit Structure will place it in the correct location.

Another reason to import a DWG file is to add details to the project that have been drafted in another CAD package. The process is slightly different but just as simple.

Importing DWG Data for Detailing

When an architectural CAD plan is placed into the model, it is usually obvious that it is going to be placed in a level. When you are placing a detail, where to actually put it is not as obvious. This is because you have to think of a detail as a supplement to a drafting view. In other words, the CAD detail is going to be placed in a view that is probably nonexistent. You need to create one by choosing View ➢ New ➢ Drafting View. Or you can select the View tab on the Design bar and click the Drafting View button. Next, as shown in Figure 3.11, a dialog box will appear asking you to name the new drafting view. You are also prompted to specify a scale. Of course this is a DWG, so you still need to deal with the scale. If you do change it, don't worry. You can always change it to something else once the file is imported. It is nice to get it "in the ballpark," though.

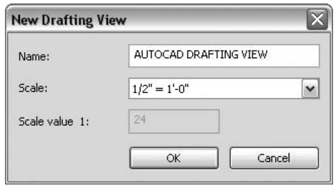


Figure 3.11
The New Drafting View dialog box allows you to create a blank view in which to insert your DWG.

Once you are in the view, you can see that it is a completely blank canvas. You can always go to the Drafting tab of the Design bar, and use Revit Structure lines to draft away. Of course, you can import a DWG here as well. If you select File ➢ Import/Link ➢ CAD Formats, you can browse for a DWG. The choices will be the same except for one that is grayed out. Not available are Current View Only and the Place At choice, as shown in Figure 3.12.

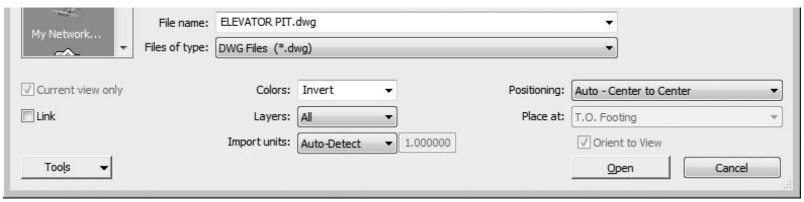


Figure 3.12
The Import dialog box for importing into a drafting view

Given the nature of this import, you will probably not want to link the DWG. The positioning is going to work using Auto – Center to Center. The colors can be black and white because it is known that you are going to want to plot this directly. You also need to watch out for the units. Be sure to take a measurement of a known length after the import just to make sure you didn't import it as the wrong scale. Once you click OK, the DWG will be imported. If you do not see it, you can press ZA on your keyboard and it will come into view. If your line weight settings have been configured correctly, it will come in with the correct line thicknesses, as illustrated in Figure 3.13.

There is much more that you can do with DWGs after they are brought into Revit Structure. The layers can be manipulated, deleted, or even halftone.

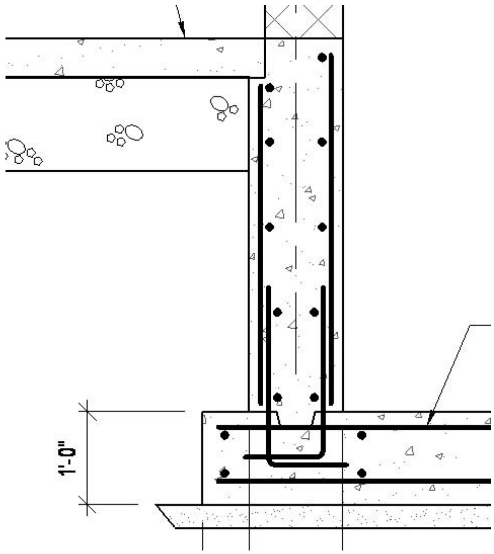


FIGURE 3.13
The imported DWG should look exactly the way you want it. If not, you need to configure your import/export line weight settings.

Controlling the DWG's View Settings

Layers have always been a necessary evil throughout the years. You don't think of the phrase "CAD standards" without the word "layers" jumping into your mind. Some firms have wonderful layer management; others, not so wonderful. Either way, once DWG files become referenced into one another, you still wind up with a huge convoluted list to search through. Revit Structure handles the import of layers differently. The program will actually group the layers with their corresponding imported DWG file. This is perfect! We do like layers when they can be separated so we can see what we are looking at.

There are two different ways to view and to manipulate an import's layers. One way is through the Visibility/Graphic Overrides dialog box of the specific view. Say, for instance, you want an architectural underlay to be halftone. You can simply open that view, and press VG on your keyboard. This will open the Visibility/Graphic Overrides dialog box for that view. If you click the Imported Categories tab, you will see the DWGs that are imported into the model, as shown in Figure 3.14.

If you select the Halftone check box, it will automatically shade the entire DWG. If you expand the tree adjacent to the DWG name, you will see all of the imported layers associated with the DWG. You can simply turn them on or off as needed for each item. Once you are done, click OK, and the import is changed as shown in Figure 3.15.

Another way to manipulate the DWG, along with its associated layers, is to select the imported instance and look at the Options bar. You are given some choices, as you can see in Figure 3.16. If the file is linked, some of the choices will be grayed out. If the file is imported, you can explode it into Revit Structure line elements.

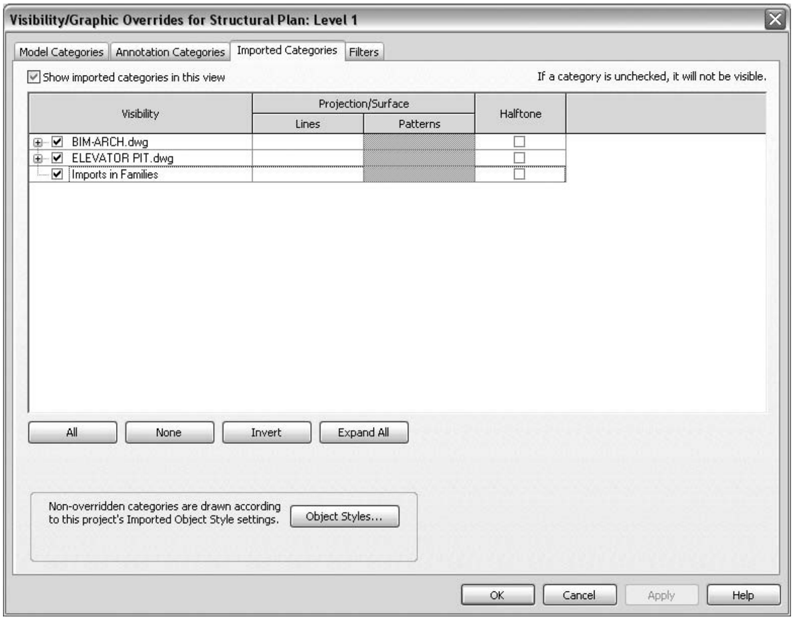


FIGURE 3.14
The Visibility/Graphic Overrides dialog box for the plan view allows you to access the layers for the DWG import. Press VG to access the dialog box.

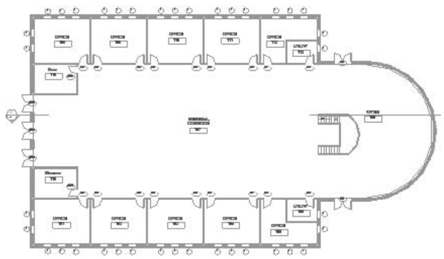


FIGURE 3.15
Automatic shading of architectural underlays is accomplished in the Visibility/Graphic Overrides dialog box.

FIGURE 3.16
When the DWG import is selected, the Options bar populates with applicable choices.

The first button is Delete Layers. Once you click this button, a list of layers will appear. You can delete any layer by simply checking it, as shown in Figure 3.17. Be careful here because this is not like AutoCAD, where you can only "purge" unused layers. Revit Structure will delete any layer you choose permanently, along with the elements that are on those layers.

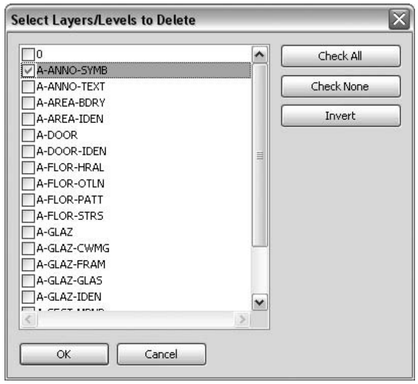


FIGURE 3.17 The Select Layers/Levels to Delete dialog box should be used with caution. Once you delete the layer, you are also deleting the elements on those layers.

The next available choice (if the file is linked) is the Query button. Once you click this button, you can select any object in the imported DWG, and a dialog box will appear providing descriptive information, and letting you specify how you would like to handle the item (see Figure 3.18).

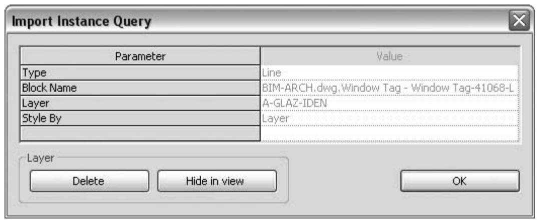


FIGURE 3.18 The Import Instance Query dialog box allows you to investigate the item and deal with it as you like.

Following the Query option are the good old-fashioned Draw Order options. They give you better control over the interference the DWG causes when laid under Revit Structure objects. You can also set the DWG as the background or foreground here, which will prevent you from accidentally selecting the DWG as you work in the Revit Structure model. See Figure 3.19.

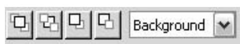


FIGURE 3.19 You can change the draw order of the DWG from the Options bar.

If you select an imported DWG that was not linked, you will have two more choices, Full Explode and Partial Explode, as shown in Figure 3.20. If you perform a full explode on the item, it will "dumb" the DWG down to Revit Structure line elements, including any embedded blocks. If you perform a partial explode, it will explode the item to line elements as well, but it will not explode embedded blocks.

FIGURE 3.20
If the DWG is not linked, you get two additional options: Full Explode and Partial Explode.

As you can see, the ability to bring in and manipulate a DWG file is something that makes Revit Structure outstanding in terms of time-saving and accuracy when it comes to sharing coordinates. The following exercise will walk you through the steps of importing a DWG as an architectural overlay.

EXERCISE: ADD THAT CAD!

This exercise will guide you in the placement of an AutoCAD DWG reference file. The file is intended to be used as an architectural overlay to start modeling your plan.

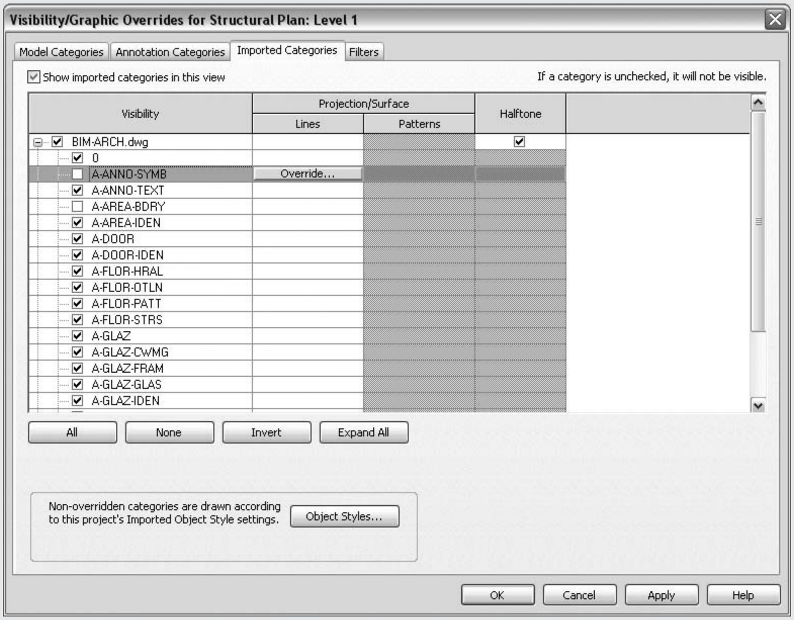
1. At the book's companion web page (`www.sybex.com/go/masteringrevitstructure2009`), find the file called `BIM-ARCH.DWG` and save it on your local drive or a network. Or you can use you own AutoCAD file if you have one.
2. Start a new default Revit Structure model, and go to Level 1 in the Project Browser.
3. Choose File ➢ Import/Link ➢ CAD Formats.
4. Browse to and select the `BIM-ARCH.DWG` file.
5. Select the option Current View Only.
6. Specify that you want to link the DWG.
7. Assign Colors: Black and White, Layers: All, and set Import Units: to Inch.
8. Set the positioning option to Auto – Origin to Origin.
9. Click Open.

This is what you need to do when you bring in a CAD file. As mentioned earlier, be deliberate with the choices made here. Most of them cannot be undone.

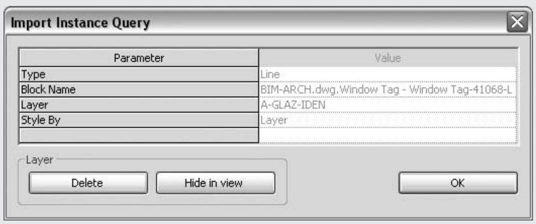
The next set of steps will allow you to manipulate the DWG:

1. Press VG on your keyboard to open the Visibility/Graphic Overrides dialog box.
2. Click the Imported Categories tab.
3. Click the Halftone button for the BIM-ARCH import.

4. Expand the layer list and deselect A-ANNO-SYMB.



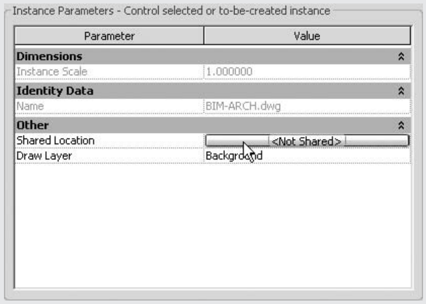
5. Click OK to close. Select the DWG.
6. On the Options bar, click Query.
7. Click one of the window tags.
8. Click Delete in the Import Instance Query dialog box.



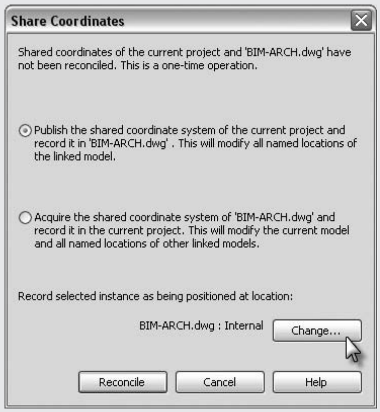
Now that the DWG is in place and looking good, it is time to nail down the positioning. Note that you do *not* have to share coordinates in every project, but being familiar with this method can be a lifesaver.

1. Select the DWG object.
2. On the Options bar, click the Element Properties button.

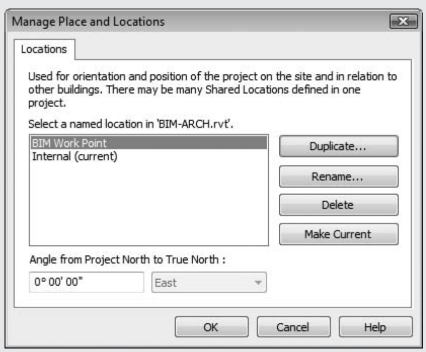
3. Click the Not Shared button next to the Shared Location parameter.



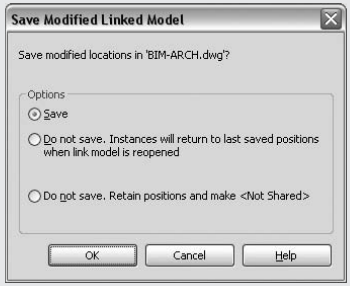
4. Because you have not shared any DWGs in this project yet, choose the first option, to publish the shared coordinate system of the current project and record it in `BIM-ARCH.DWG`.
5. Click the Change button under Record Selected Instance as Being Positioned at Location.



6. Click the Duplicate button.
7. Call the new button **BIM Work Point**.
8. Click Make Current.



9. Click OK.
10. Click Reconcile.
11. Click OK.
12. Save the model.
13. Click Save in the Save Modified Linked Model dialog box.



To reiterate, sharing the coordinate system between AutoCAD and Revit Structure is not always necessary, but this question is bound to come up sooner or later in a project. It is beneficial to know how to handle it when the time comes.

Of course, linking CAD is one way to start a project. But what if the architect is using Revit Architecture? Then you can take full advantage of BIM!

Linking Revit Architecture

This is what it's all about here! When the moons are aligned, and the project team is up on Revit Structure across the board, the building will just model itself. No? Okay, there still needs to be intervention from design professionals, but the coordination can be a real lifesaver. And it's easy for the most part. Luckily the concept is similar to linking in a CAD format; only the link is assumed. Note that if you are sending linked models to other firms, you must include all of the models involved, similar to the external referencing system within AutoCAD and MicroStation. Once the Revit Structure models are linked, however, is where the similarities end. In the linked Revit Structure models, you will have the opportunity to physically copy objects from one linked model to the current one. Along with this copy, you can choose to monitor the movement of the architectural model elements each time you reload it. Almost as important, you can perform collision detection between the two models throughout the life of the project.

To begin our example, we create a blank Revit Structure model and link a partially completed Revit Architecture model. To add the linked model, choose File ➢ Import Link ➢ Revit. In the Import/Link RVT dialog box, you will see at the bottom that the only choice you need to make involves the positioning. Of course, by now you realize that you cannot use shared coordinates because nothing has been shared yet, so the choice for this example is Origin to Origin as illustrated in Figure 3.21. This means that the internal coordinate system will match the internal coordinate system of the Revit Architecture model.

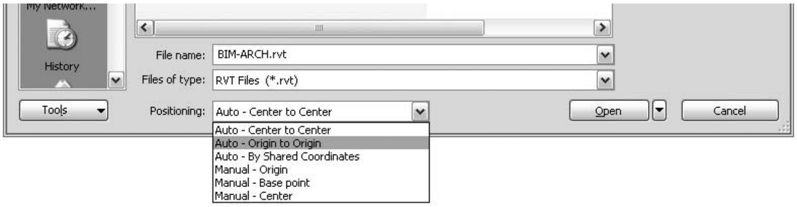


FIGURE 3.21
Choosing the Origin to Origin option upon import / link.

Once you click Open, the architectural model will load—maybe. There are a few settings you need to check before you can actually see the architectural model in its entirety. The first thing to look for is the View properties. By pressing VP on your keyboard or by right-clicking and choosing Properties, you can access the View Properties dialog box. The reason you may not be able to see much of the model is the fact that the view, by default, is probably set to Structural. Since you are trying to coordinate with a Revit Architecture model, you must set the view's Discipline to Coordination under the Graphics category, as shown in Figure 3.22.

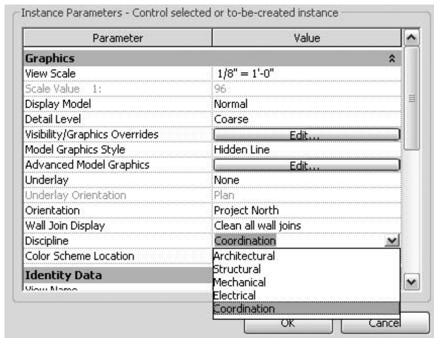


FIGURE 3.22
To coordinate with a Revit Architectural model, change the Discipline setting to Coordination.

The reason you cannot see even the architectural walls is because in the Revit Architecture model, they are set to Non-bearing. Some of them should be non-bearing, but some of them should be shear walls, which is why the architect hired the structural engineer in the first place!

DISPLAYING THE REVIT ARCHITECTURAL MODEL

There are some items in the architectural model you may want to see, and others you may not. Also, if you do want to see some items, you have several choices in terms of manipulating the architectural link to suit your needs. The first method will be setting the visibility graphic overrides. Pressing VG on your keyboard opens the Visibility/Graphics Overrides dialog box for that view. Click the Revit Links tab. Now you can see the check box that allows you to halftone the link, as shown in Figure 3.23.

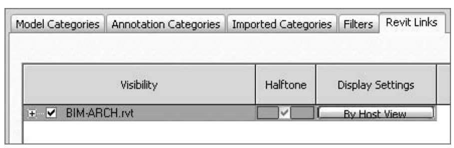


FIGURE 3.23
The Visibility/Graphics Overrides dialog box allows you to access the display settings of the linked architectural model.

On the right side of the dialog box, you will see a button under Display Settings. If you click the button, the dialog box splits into four tabs. The Basics tab allows you to control the settings By Host View (the default), By Linked View, or Custom, as you can see in Figure 3.24. By choosing Custom, you can change any graphic in the architectural model you wish. This will *not* affect the actual linked file.

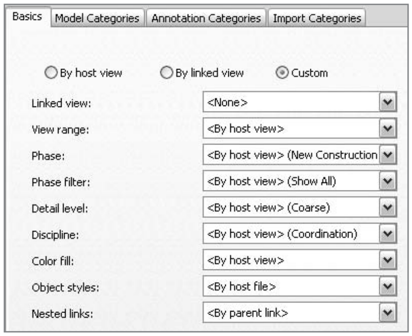


FIGURE 3.24
By changing the visibility graphics for the view, you can manipulate the graphics for the linked file any way you choose.

The next tab is the Model Categories tab. If you chose Custom, you will have access to every model component. This allows you to alter the appearance of the linked architectural model just as if it was physically constructed within the structural model.

The Annotation Categories tab allows you to change textual items, dimensions, and tags separately from the model categories, again just as you would in the structural model.

The Import Categories tab lets you reach into the linked file and control the graphics of any imported or linked files that reside within the architectural model.

Another good way to manipulate the visibility of the Revit Structure underlay is to simply hover your mouse over a specific item in the import that you want to change graphically, and then press the Tab key on your keyboard. This will highlight the item. Once the item is highlighted, you can right-click and select either Hide in View or Override Graphics in View, as Figure 3.25 shows.

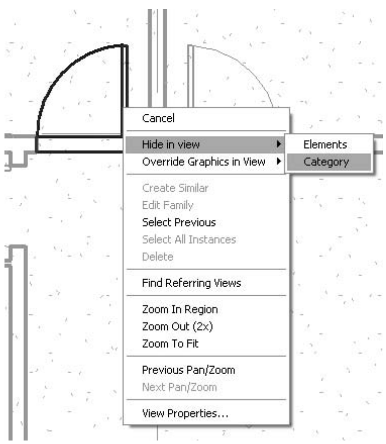
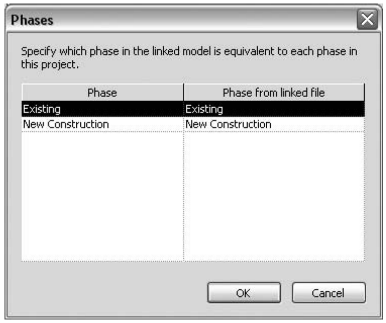


FIGURE 3.25
By Tab-selecting specific items in the Revit Structure link, you can control the graphics element by element.

In terms of phasing, this entire link can be applied to a phase in the model. If you select the link and click the Element Properties button on the Options bar, you can click Edit/New to access the type parameters. There, you can locate the Phase Mapping variable and click the button to its right. In the dialog box shown in Figure 3.26, you can map the phases within the Revit Architectural model to the phases within the Revit Structure model.

FIGURE 3.26
The Phases dialog box lets you specify which phase in the linked model is equivalent to each phase in your project.

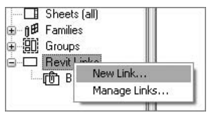


Of course, once the link is established, the architect has probably not stopped working. Revit Structure provides methods in which you can manage the linked Revit Structure models.

MANAGING THE REVIT ARCHITECTURE LINK

Similar to an external reference, Revit Structure keeps a live path back to the Revit Architecture model. This link can be managed from the Project Browser. Once you have established a link in the model, you can then right-click the Revit Links category in the Project Browser, as shown in Figure 3.27. If you are about to add a new link, select New Link to open the browser, which allows you to choose a Revit file to link.

FIGURE 3.27
The best way to manage linked Revit Structure models is to use the Revit Links category in the Project Browser.



If you choose Manage Links, the Manage Links dialog box opens. The default tab is called CAD Formats. If you had a DWG or DGN linked in, you would see it listed here. If you click the Revit tab, you will see the Revit link(s) available, as shown in Figure 3.28. If you then select a Revit link, you can modify some of the options available on the row. The first option lets you specify whether the link is an overlay or an attachment.

Overlay or Attachment is in accordance with the Type parameter. If you choose Overlay, Revit will *not* load any nested linked models. If you choose Attachment, Revit *will* load any nested linked models. Of course, this option can make a huge difference in the size of the file!

The next choice, Saved Path, corresponds directly to the control to the right of it, Path Type. Figure 3.28 shows a relative pathing scheme. This means that the entire path up to the filename is a wildcard. If the two files are in the same directory, the folder names of that directory can change at any time. Revit Structure will know to look only for the RVT file with the identical name. With this pathing, the files must at least be in the same directory. This is ideal for sending Revit Structure models with links to clients. All clients need to do is copy all the files in the same directory, and the links will reestablish. If you choose an absolute path, the entire path all the way back to the drive letter is required. If any one of the folder names change, or if the file is relocated, the path is broken and Revit Structure will not be able to find the link.

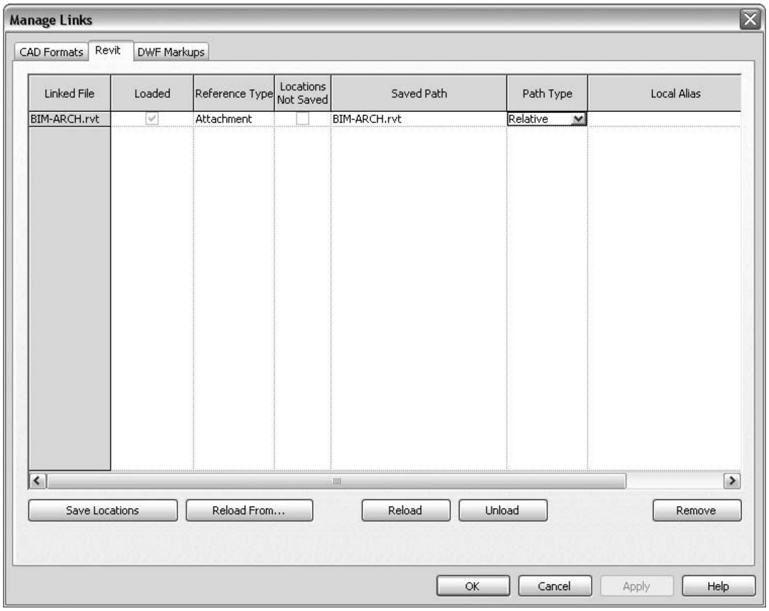


FIGURE 3.28
Choose File ➢ Manage Links, or simply right-click the Revit Links category in the Project Browser, to access the Manage Links dialog box.

With the link still selected in the Manage Links dialog box, you'll see a row of controls across the bottom of the dialog box, which allow you to perform tasks specific to the link (see Figure 3.29). The first button, Save Locations, allows you to force the location to be saved. This will establish a stronger path if the files are moved. Revit Structure warns you that if you are planning on sending this model out of the office, you must not click the Save Locations button. The recipient of the model will have to deal with this obviously unrelated path.

Reload From is a nice feature because you can keep the same link with the same coordinate location while loading different underlays for comparison. If you do load from another location for comparison purposes, be sure to load the correct model back when you are done.

Of course, you can reload, unload, and remove the link altogether with these tools. Many times when the file is getting large, simply unloading links that are not used all that much can greatly improve your performance.

FIGURE 3.29
With the link selected in the Manage Links dialog box, you can perform several functions related to how the links will be read into the model.

Also, a few of these choices are available without even going to the Manage Links dialog box. As you may notice, the Project Browser's Revit Links category will display the available links. If you expand the Revit Links category, you can right-click on the link itself and access the same commands as found in the Manage Links dialog box, as shown in Figure 3.30.

There are two additional choices available when you right-click on the link in the Project Browser. One is Create Instance. You can essentially copy the link. Revit Structure will still only consider that there is one externally linked model. Any changes to the architectural model will be reflected in both links.

The other choice is the Match option. This is handy when you have gone through painstaking efforts to get the link displayed exactly the way you like. If you have another instance of the link, you can simply match the "good" link and change the display configurations of the "bad" link.

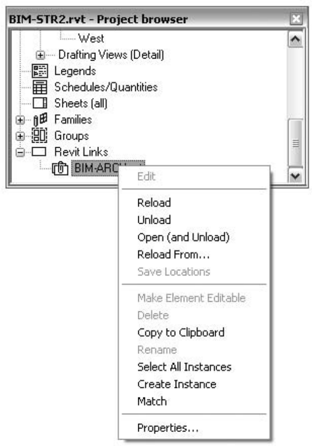


FIGURE 3.30
By right-clicking directly on the link in the Project Browser, you can manage the link "on the fly."

One other way to remove a link is to simply delete it. Revit Structure will give you a warning, as shown in Figure 3.31, that says "All instances of Linked Model 'BIM-ARCH.0003.rvt' have been deleted, but the file itself is still loaded. Remove the Linked File using Manage Links dialog to save memory unless you are going to reuse it in this project. Removing the Link cannot be undone." If you click Remove Link, the instance is cleansed from the model.

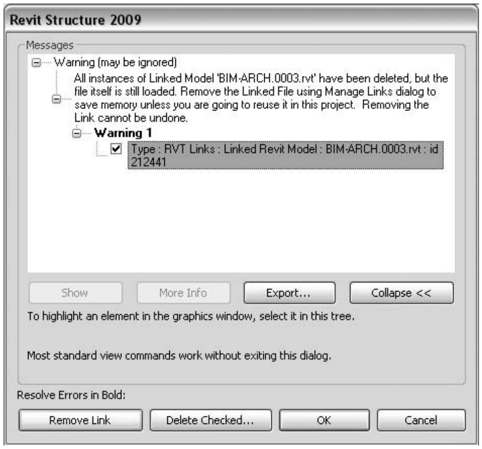


FIGURE 3.31
Select the linked Revit Architecture model and delete it. Revit Structure will allow you to either unload it or remove it completely.

Now that the link has been established, and you know exactly how to manage it in terms of location on the network as well as the coordinate location in the model, it is time to see how to copy some of the useful items into your structural model and to keep a live monitor on the copied elements.

COPY/MONITOR

Next comes Revit Structure's shining moment: the ability to copy in elements from a linked architectural model and then monitor the movement of those elements is what BIM is all about. But hold on for a moment—although this is a great feature, it is at this point you need to remember that Revit Structure is simply a tool, a tool to aid you in your design process and workflow. Yes, if the architect changes an element you are monitoring you will be notified, but just remember to check what those changes do to the model. As you know, in Revit Structure a change in one place will trigger changes in other places. A computer application will not replace the human eye.

Here's how it works. Once the Revit Structure link has been established, the next step in the structural design process is to basically "draw over" the top of the underlay. If that underlay has crucial structural elements you need, you can copy them. It's as simple as that. Once they are copied, they can be linked. In this example, an architectural model has been linked into Revit Structure. The column grids have already been laid out in the architectural model. On the Tools toolbar, click the Copy/Monitor button. Once you do this, you will see two choices, as shown in Figure 3.32. One is Use Current Project (for multiple links), and the other is Select Link. If you choose Select Link, you can then select the link you want to harvest the grids from.

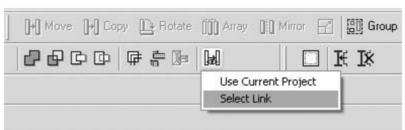


FIGURE 3.32
Select the Copy/Monitor button to start the process.

The Design bar now includes only one tab: Copy/Monitor. Notice there are a few commands available, as you can see in Figure 3.33. You can click the Copy button, and Revit Structure will allow you to select the items you want to copy. These items can be monitored as well.

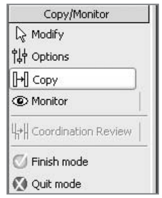


FIGURE 3.33
The Copy/Monitor tab on the Design bar has tools to aid you.

When you click the Copy button, Revit Structure will display your options on the Options bar. If you are going to select more than one item, it is important that you enable the Multiple check box, as shown in Figure 3.34. This tells Revit Structure to keep a running tally of the items being copied. One thing to be careful of is the fact that, even though you checked the Multiple option on the Options bar, you still need to press and hold the Ctrl key to select the multiple items. Once you have finished, click Finish on the Options bar (see Figure 3.34).

FIGURE 3.34
Be sure to click Multiple, then Finish when you are done selecting the items you want to copy. If you don't, you will have to go back and remake the selection set.

Usually once you finish copying the items, you will see a Duplicate Types dialog box informing you that the items you are copying into the Revit Structure model match the ones from the linked model. Revit Structure will keep the items in the current model. It is safe to click OK in this dialog box.

Once this process is finished, the eyeballs appear, as shown in Figure 3.35. This indicates that the items have been copied and are being monitored successfully. On the Copy/Monitor tab, click Finish Mode, and the new items are now in your model.

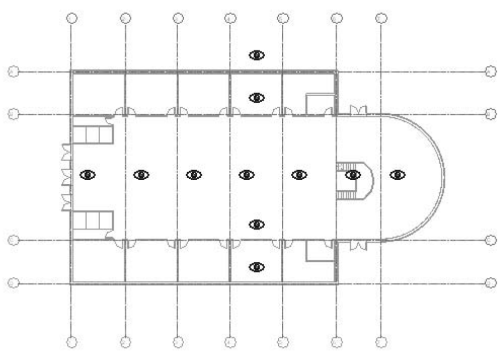


FIGURE 3.35
A blue eyeball symbol indicates a successful monitor on a given object. Once you see this verification, it is safe to click Finish Mode on the Design bar.

Coordination Alert

Now for the fun stuff. When the architect changes one of the items that have been copied, you will be notified as soon as you reload the link. Of course, if changes were made to the linked model when you did not have your model open, you would get the alert upon opening. Be careful because you will only see a warning, as shown in Figure 3.36, and it is easy to dismiss the warning and go back to business, forgetting that the link now needs a coordination review.

FIGURE 3.36
When your model is reopened, or the Revit Structure link has been reloaded, you get a coordination warning.

You can simply close the warning and there is no longer an indication something is amiss. But if you select the Revit Structure link, you will see a button with an exclamation mark in the Options bar. This is the Coordination Review button. Click this button, and the Coordination Review dialog box opens, as shown in Figure 3.37.

In the Coordination Review dialog box, you will see the item in question. In this example, it is a grid. Once you expand the tree, you can see exactly what is different between the two models. In addition, you are given four options: you can choose to postpone, accept, reject, or modify your grid. If you choose the Modify option, Revit Structure will change your grid to match.

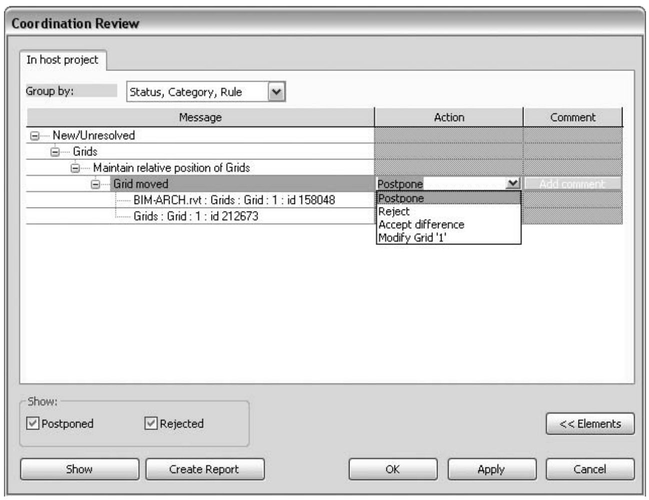


FIGURE 3.37
The Coordination Review dialog box allows you to modify your item to match the linked item.

Also, you can create a report that will list exactly what occurred in this session by clicking the Create Report button at the bottom of the dialog box. The output of that report will be

in HTML format so you can easily post it to a file-sharing site, Internet/intranet, or a local file server. Click OK in the dialog box, and the grid is changed.

Monitor

At times, you may already have an element in place in your Revit Structure model that you wish to just monitor against the linked model. This can be accomplished in a similar manner to the Copy/Monitor method. In this example, the view will be switched to an elevation. A Level 1 and a Level 2 already exist in the active model. Level 2 is 2′-0″ lower than the linked model. This is fine if you want the gap to remain. Revit Structure will still allow you to monitor the movement of an offset element. If you click the same Copy/Monitor button, you can select the link. Once you do this, the Design bar will switch to Copy/Monitor mode. Now, instead of clicking Copy, you can simply click Monitor, as shown in Figure 3.38.

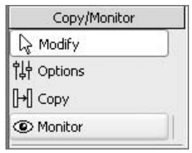


FIGURE 3.38
By clicking Monitor, you can keep an eye on the architectural element without having to copy it.

With the Monitor command running, you first select your element, then click the linked element you are monitoring against. Again, you should look for the blue eyeball as an indicator that you have successfully selected the two items and have a working monitor. Once you have finished with the monitoring elements, you can click Finish Mode. Now your models are monitored.

Real World Scenario

MAINTAINING A STEEL-TO–FINISH FLOOR RELATIONSHIP

Comparisons have been made between the differences of setting the structural levels below the architectural levels by the difference of the floor slab + the finish, and offsetting your framing by that same amount. It was found that it is much easier to maintain top of steel elevation when you are using a level that is true to the top of steel.

Copy/Monitor Options

There are always additional options to every Revit Structure command, and Copy/Monitor is no exception to that rule. As you engage the Copy/Monitor command, you will notice an Options button on the Design bar. If you click it, you will launch the Copy/Monitor Options dialog box. In this dialog, you will see five tabs along the top. These tabs categorize the model into component types. This dialog box allows you to replace linked items with model items as you copy the elements into your model. It is *always* a good idea to examine this dialog box. Say you want

to copy items such as walls; by default, Revit Structure will substitute the linked wall with a generic 8″ wall, as shown in Figure 3.39.

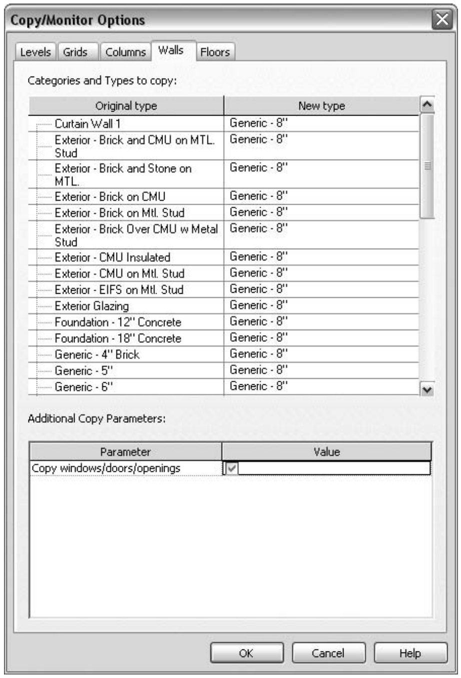


FIGURE 3.39
Always check the Copy/Monitor Options dialog box if you are copying/monitoring more than just annotation elements.

With this task complete, you can now start to do some reconnaissance.

INTERFERENCE CHECK

Linking models together allows you to see where collisions are occurring between the linked files. In Figure 3.40, a common collision between stairs and a beam flange has been detected.

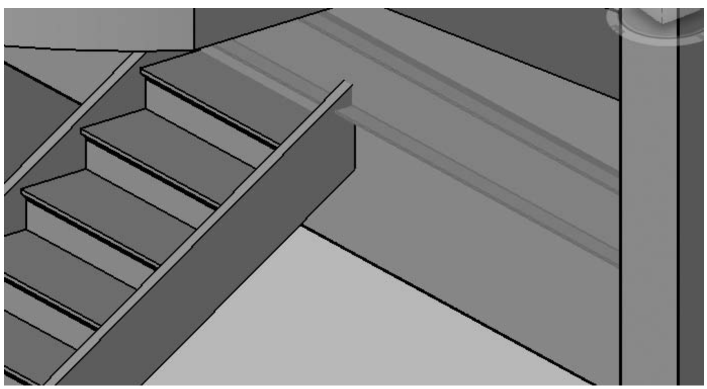
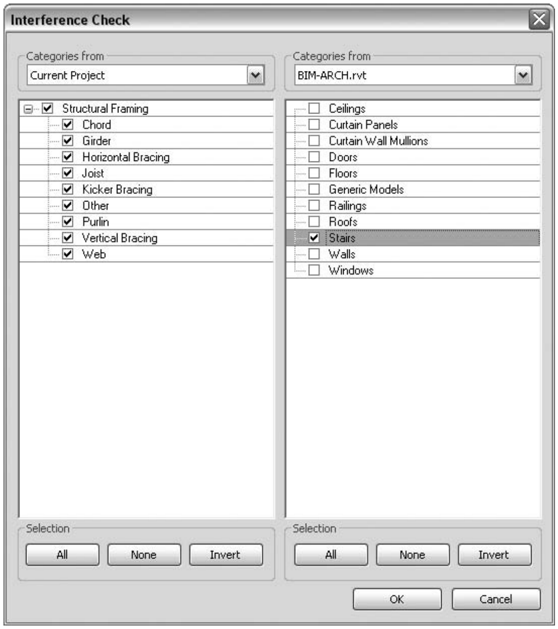


FIGURE 3.40
Collision detection will highlight the items that are colliding.

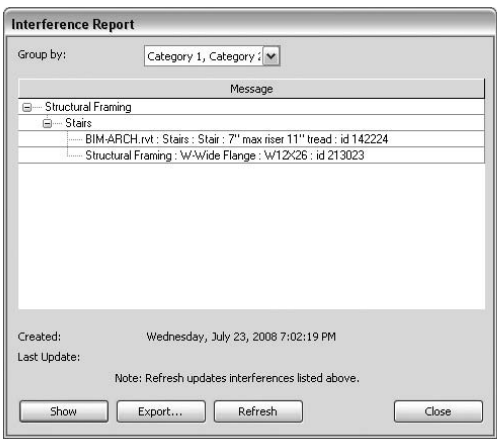
To run the Interference check, select Tools ➢ Interference Check ➢ Run Check. The resulting Interference Check dialog box contains two panels, as shown in Figure 3.41. In the left panel, the current model is selected from the menu at the top of the panel, and we enabled Structural Framing. In the right panel, the linked model is selected and we chose Stairs for the clash detection.

FIGURE 3.41
The two models can be compared side by side.



Once the suspected items are checked, click OK. Revit Structure will find all of the clashes between structural framing and stairs. In the Interference Report dialog box, you can select the items that appear in the report, as Figure 3.42 shows. The items will then be highlighted in the model. If you cannot see the items, click the Show button at the bottom of the dialog box. Revit Structure will zoom to a reasonable distance in an existing view to allow you to see the collision.

FIGURE 3.42
The Interference Report allows you to graphically review each collision.

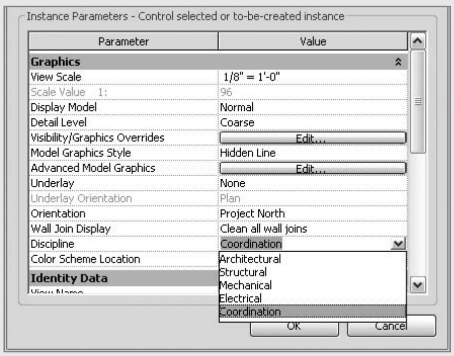


Unlike the Coordination Report, with the Interference Check Revit Structure will not take an action to fix the solution—you have to do it manually. Once the issue is fixed, you can refresh the report, and Revit Structure will clear the field, thus giving you a clean interference report card.

Exercise: Linking the Architecture

This exercise will involve linking and then performing a Copy/Monitor operation to the architectural underlay. You do not need Revit Architecture to complete the exercise. At the end, to test the coordination you can close the structural model and then open the architectural model directly in Revit Structure. Make some changes to the architectural model, close it, and then open the structural model to review the report.

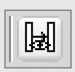
1. Copy the file called `BIM-ARCH.rvt` from the book's web page to a convenient location. (You can use your own file for this exercise as well.)
2. Start a new default Revit Structure model.
3. Go to Level 1 plan view.
4. Select File ➢ Import/Link ➢ Revit.
5. Browse to the `BIM-ARCH.rvt` file.
6. Set the Positioning to Auto – Origin to Origin.
7. Click Open.
8. Press VP on your keyboard.
9. Set the Discipline to Coordination. Click OK to close.



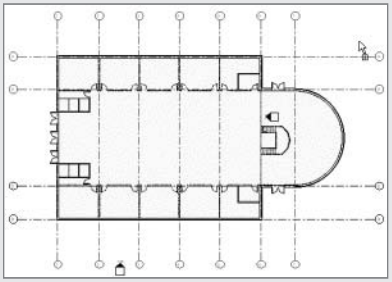
10. Go to a 3D view.
11. Press VP again. Set the 3D view's Discipline setting to Coordination. Click OK to close.
12. Go back to Level 1 plan view.

See how easy it is to link Revit into Revit? The next series of steps will lead you through the Copy/Monitor functionality.

1. On the Tools toolbar, click the Copy/Monitor button.



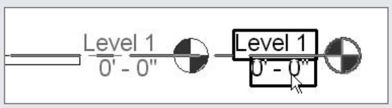
2. In the menu flyout, click Select Link.
3. Select the architectural model.
4. In the Copy/Monitor bar, click the Copy button.
5. On the Options bar, click the Multiple check box.
6. While pressing down the Ctrl key, select all of the structural grids.



7. Once you have finished, be sure to click Finish on the Options bar.
8. Click OK in the Duplicate Types dialog box.
9. On the Copy/Monitor bar, click Finish Mode.

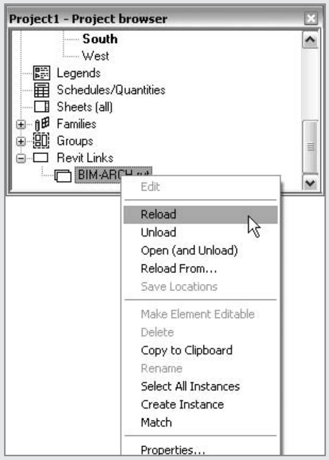
Now that you have copied and monitored the grids, it is time to monitor some levels in the structural model.

1. Go to the South elevation.
2. Click the Copy/Monitor button on the Tools toolbar and choose Select Link.
3. Select the linked model object.
4. In the Copy/Monitor tab, click the Monitor button.
5. Select the Level 1 that is in the structural model.
6. Select the Level 1 that is in the linked file.

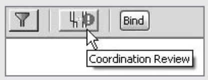


7. Repeat the procedure for Level 2. This time there is a gap between the two levels.
8. On the Copy/Monitor bar, click Finish Mode.
9. Open the linked model called `BIM-ARCH.rvt`. You can keep your current Revit Structure session open and just open another session of Revit Structure, and then open the `BIM-ARCH.rvt` model.
10. In the Architectural model, move Grid 1 2´-0˝ to the right.
11. Save and close the architectural model.
12. If you have closed it, open the structural model. You will see a coordination warning. You can dismiss the warning.

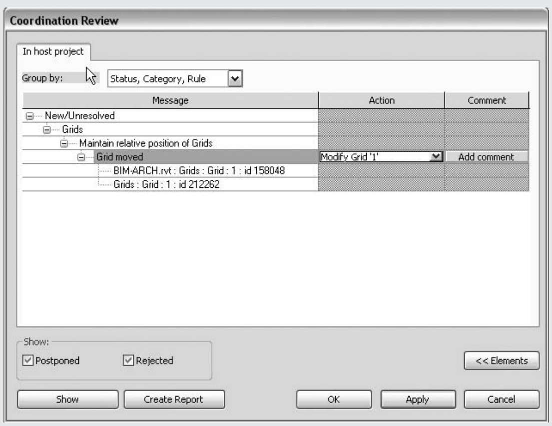
13. If you have not closed it, right-click on the `BIM-ARCH.rvt` link in the Project Browser and select Reload from the menu.



14. When you see the coordination alert dialog box, close it.
15. Select the architectural link.
16. On the Options bar, click the Coordination Review button.



17. Expand the Grid Moved tree, and click between BIM-ARCH and Grids. If it is visible, the grids in question will be highlighted.
18. For the Action option, select Modify Grid '1'.



19. Click OK to close.

The first portion of this chapter illustrated how to copy grids from an architectural model. Revit Structure has the ability to add grids and elevations without the use of an architectural source. Next we'll focus on adding structural grids and elevations from scratch.

Creating a Structural Grid

You can create a structural grid in Revit Structure, and the process is surprisingly painless to boot! It basically goes like this: you use the Grid command to draw the grid by either selecting two points, or by selecting an object or reference plane. There is no wrestling match with having to specify the entire grid or convoluted dialog boxes that continuously pop up. Also, the grids will number or letter themselves sequentially.

For our example, we will begin with a blank model. By default, two levels are available. Normally, it is good practice to have all of your levels created before the grid layout is created, but in this example we want to allow Revit Structure to behave in a certain way so we can examine the procedure to "fix" some things. As it stands, if the grid is in place first, and a new level is added above or below existing levels, the grid won't reach that level unless you perform a function called Propagate Extents. Propagate Extents will be reviewed later in the chapter.

To begin the first grid, select the Basics tab on the Design bar and click the Grid button, as shown in Figure 3.43.

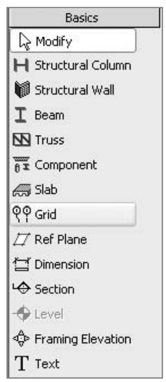


FIGURE 3.43
On the Basics tab of the Design bar, you can start the Grid command.

Once you click the Grid button, the Options bar populates with controls. By default the initial choice is to draw the grid by selecting two points in the active plan view, as shown in Figure 3.44. Once you select the two points, you can see that Revit Structure labels the grid. This label may not be the one you want. If you want this label to be correct, and the subsequent labels to be sequential based on the correct label, you need to change how the grid head reads before you add the next grid.

If the first grid is grid 1, then the next grid will be grid 2. To add the second grid, start the Grid command again. This time, you can click the Pick button on the Options bar, as shown in Figure 3.45. Once you do, you can add an Offset distance. This example has a distance of 25′-0″. All you need to do is select the first grid. As you are clicking, be careful, and watch for the alignment line. If it is heading in the right direction, you can click the existing grid. The new grid is created 25′-0″ away from the selected grid, and it becomes number 2.

Figure 3.44
Add the grid by selecting two points.

Figure 3.45
The Grid Offset control can be used to create new grids an offset distance from existing elements.

You can keep selecting grid after grid to create subsequent grids. A nice thing about the command system in Revit Structure is that the command keeps running until you stop it. If you need a spacing other than 25′-0″, you don't have to leave the command to add a new increment. Simply change the spacing, and offset the next grid. It could not be easier.

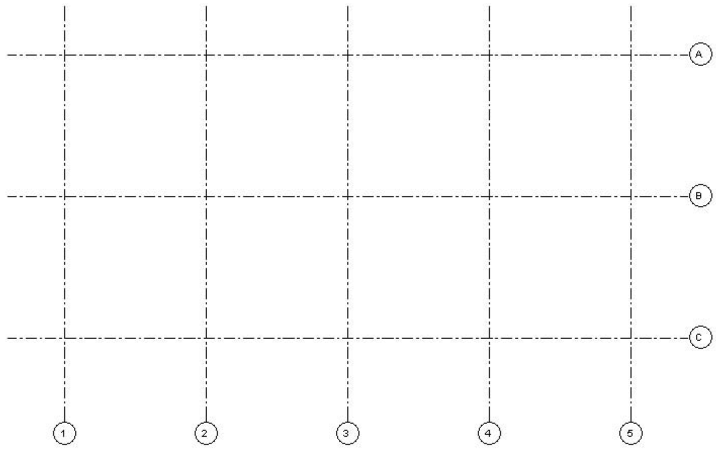
Why Is This Number Used?

A common mistake when adding grids revolves around the fact that command keeps running. When you first start using Revit Structure, you may find that when you are adding grids, you will forget the command is still running. This always happens to those who are new to Revit Structure. The common mistake is an individual will go to rename the grid and accidentally add the next grid directly on top of the existing one, thus duplicating the grid. This will mask one of the grids. Then when the individual wants to relabel another grid, Revit Structure does not allow a duplicate name. With a more complicated grid system, it can be difficult finding where the obscured grid is hiding so you can delete it.

Adding a grid in the perpendicular direction is just as easy. Click the Grid button, and draw the grid in the other direction crossing the other grids, as shown in Figure 3.46. The clever thing here is that not only are you adding a new gridline, but you are also creating grid intersections. When the columns are placed along these lines, they will move with the grids and will display the grid location for certain types of schedules.

The first new direction grid label should be renamed. Be sure you are not duplicating the grid. Change the name, and start the Grid command again. You can now work with the Options bar to add several rows, as indicated in Figure 3.46.

FIGURE 3.46 Adding a perpendicular set of grids is the same procedure. Be sure to label the first grid of this series.

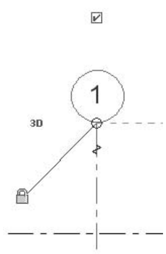


Modifying the Gridline

As you add the grids, you may notice that the grid head may not be on the correct side. You may even want a grid head on both sides. Also, you need to make adjustments when you get into a tight situation where the grid head has to "jog" to the side.

To adjust a grid, make sure you are not still in the Grid command, and select a grid object. A few blue grips display, as you can see in Figure 3.47.

FIGURE 3.47 By selecting the grid, you will gain access to controls allowing you to graphically modify each individual grid.

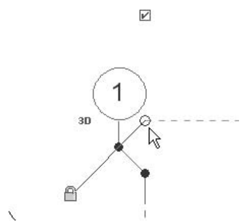


The first item is a check box that allows you to turn the grid head on or off at that end. If the head is on, it will expose a jog symbol. If you click the Add Elbow icon, the gridline will have a jog added to it, and the head will become offset. This is crucial to trying to jam several grid heads into a tight spot.

Another control you can use is the alignment line across the top of the gridline. Attached to this alignment line is a blue drag circle, and attached to that is a blue lock icon. Essentially these three controls tie the individual grids to the adjacent grids. If you click and hold the blue circle, and then drag the grid end, all of the grids locked to it will move as well. If you first unlock this

grid end, it will move independently of the other grid ends. This procedure will also move the grid(s) within all other levels. However, if you click the 3D control, turning it into a 2D control, you are telling Revit Structure to ignore the other levels and only move the grid end on the current view. See Figure 3.48 for an example of a jogged grid head.

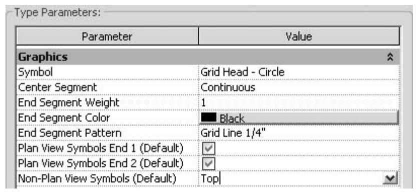
Figure 3.48
The controls shown here adjust the grid end and grid head.



Element Properties

Other changes to the grids can be made through the Element Properties dialog box. If you select a grid, you can then click the Element Properties button. In the resulting dialog box, you won't see many choices within the Instance parameters. But if you click the Edit/New button, you will discover several things that you can do. In most situations, it is good practice to click the Duplicate button; then you can change the grid head. You can even remove the center segment if you need the grid head at each end of the building but don't want to carry the actual line all the way through the building. See Figure 3.49.

Figure 3.49
The type parameters of a gridline can be altered through the Element Properties dialog box.



Type Parameters:

Parameter	Value
Graphics	
Symbol	Grid Head - Circle
Center Segment	Continuous
End Segment Weight	1
End Segment Color	Black
End Segment Pattern	Grid Line 1/4"
Plan View Symbols End 1 (Default)	☑
Plan View Symbols End 2 (Default)	☑
Non-Plan View Symbols (Default)	Top

Use a Type to Control Grid Head Display

It has been found that you will need different grid types to control the display of grid heads rather than the check box found on either end of a grid. If you create a grid using a given start and end point and want to swap the grid head to the other end, create another style with the grid head active on End 2. Odds are that your grid head end preference will be same for that grid on every plan view. Save yourself time and make that change once (via an End 2 Grid type) rather than repeating the edit work.

Radial Gridlines

The process for creating an arc grid is basically the same as adding a linear grid. To add an arc grid, click the Grid button on the Basics tab on the Design bar. In the options, click the Draw

button. For the line work options, click either the arc passing through three points, or the arc from the center and end points, as shown in Figure 3.50.

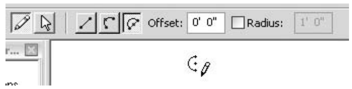


FIGURE 3.50
To add a grid arc, choose one of the arc options from the Options bar when you start the Grid command.

Once the option has been set, you can draw the arc grid wherever you need. It will behave exactly the same as a line grid, except it's an arc (see Figure 3.51).

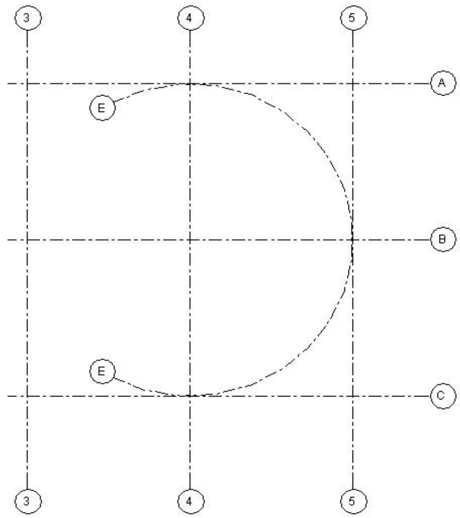


FIGURE 3.51
An arc grid that has been added to the model

The ability to freely create a column grid "stick by stick" allows you to put a grid together in a controlled approach. The same holds true for building elevations (levels). They are added to the model in almost the same manner but control much more of the model upon completion.

Creating Levels

This topic could be the most important topic in this book. What separates BIM from a referenced CAD platform is the fact that the entire model is contained within one file. If you need a horizontal roof plan, you can create one in the model. But that roof plan needs to be controlled

vertically as well. In elevation you need a datum, not only for dimensional information, but for comparison to another model you may have linked into the model.

Creating a level is an easy procedure. To begin, you must be in an elevation view. Our example will use a south elevation with the building grids in place. You typically start with at least two levels, as shown. To create a new level, select the Basics tab on the Design bar and click the Level button. When the Options bar populates, click the Pick button. Set the distance to the floor-to-floor increment, and select one of the levels. Make sure before you click on the existing level that you see the alignment line appear above that level (so you are adding an additional level *above* the level that is there already). See Figure 3.52.

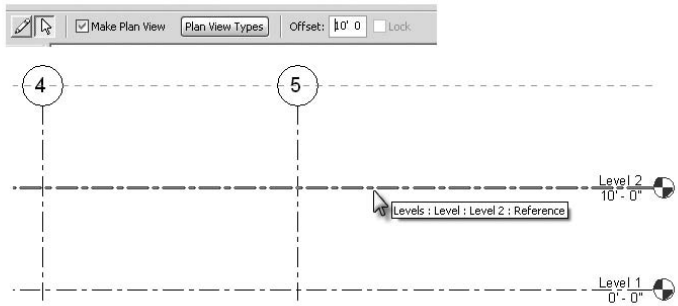


FIGURE 3.52
By using an elevation view, you can add a new level. Normally, the procedure will be to select an existing level to use as an offset for the new.

If you offset Level 2, the next level will be Level 3. You can keep offsetting levels until you are done.

With this transition complete, you can start modeling on these levels. The problem is, you have added levels but no actual floor plans exist in the Project Browser.

Creating Floor Plans

As you create a level by selecting or offsetting, you are also creating a plan view. As the levels are added, you can see in the Project Browser that new plans are being created. However, if you create new levels using Copy, a corresponding plan view is not automatically created. These new plans are directly linked to your new levels. You can control the creation of these plan views as you are adding levels to the model by paying attention to the Options bar. In the Options bar, you will see a check box that controls whether you are creating a plan view with each level. To the right of the check box, you will see a Plan View Types button. This function allows you to specify the type of plan view you are adding to the model so you can avoid having to manually configure the plan view after it has been created.

If you have been creating levels with this check box turned off, or if you have used Copy or Copy/Monitored levels into your model, you will have to manually create floor plans. Do so by choosing View ➢ New ➢ Floor Plan to open the dialog box shown in Figure 3.53. You can then choose which levels to create plan views for.

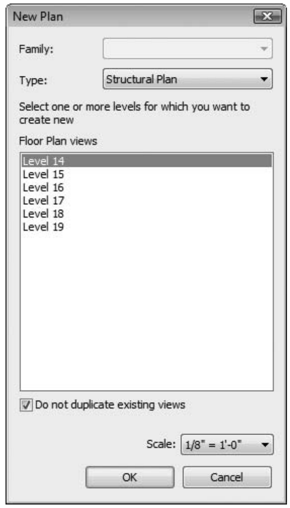


FIGURE 3.53
Use this dialog box to create any view that did not get created.

Modifying the Level Line

Just like grids, levels have similar adjustment. If you select a level, the same blue controls appear. You can turn the level head on or off by clicking the check box, as shown in Figure 3.54. You can add a jog to the level line by clicking the Add Elbow icon. It looks like a little blue jog, as shown in Figure 3.54. If you see the alignment line, this means that if you drag the level head, the rest of the levels attached to that alignment line will drag along with it.

DON'T USE THAT CHECK BOX!

Just as grids can be unwieldy by the rampant usage of the on/off check box, so too can levels. Try to plan out which end should have a level head and assign different types accordingly.

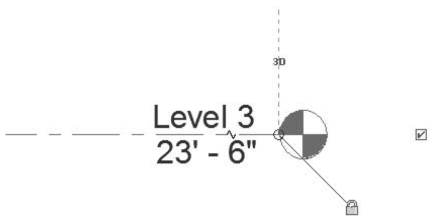


FIGURE 3.54
Select the level, and several options for modifying the level end become available.

If you click the Element Properties button on the Options bar, you can click Edit/New. Then click Duplicate and you can make changes to the level, such as changing the level head symbol.

Also, with the level selected, you see some items turn blue. This indicates that you can modify that parameter. Say you wanted the level to be at a different height. If you click the distance in the level (the 30′-0″ for Level 4) and change the number, the level will move to be that height. Also, if you do the same thing with the name, Revit Structure will rename it, but you can also rename the plan view. Before Revit Structure renames the plan view, you will see a message asking if this is what you want, as shown in Figure 3.55.

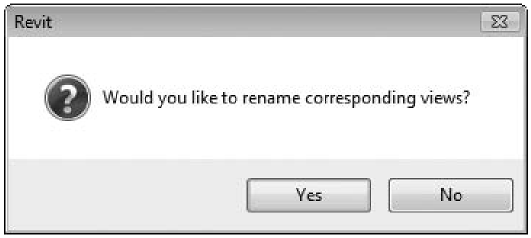


FIGURE 3.55
Renaming the level also renames the floor plan. You rename the level simply by clicking on the level in an elevation view.

Changing the Elevation

Some projects are not based at 0′-0″. Usually there is a note stating that datum elevation xxx′-xx″ = 0′-0″. But sometimes this elevation is displayed in the elevation markers. Revit Structure can handle this in three quick steps. The first step is to choose Tools ➤ Project Position/Orientation ➤ Relocate Project. Once this command starts, you are basically thrown into the Move command. Next you need to select a point along the 0′-0″ elevation line. This is the first point of the move. Move your cursor upward. You may notice that Revit Structure has a blue increment alongside your cursor, as shown in Figure 3.56.

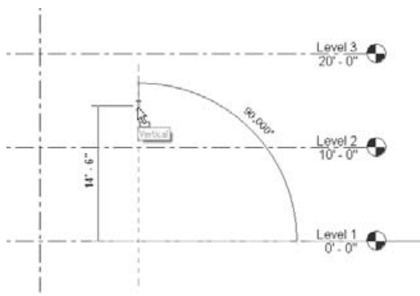


FIGURE 3.56
By choosing Relocate Project, you can move the project to a higher elevation.

For the second point, you can either select it or type the elevation. The distance you type will be your datum elevation. After you are done with this step, press ZA on your keyboard to zoom the view to be able to see the new location again. Notice that the datum elevations are still based on 0′-0″, but you can change that. Select one of the levels, click the Element Properties button on the Options bar, and then click Edit/New. In the type parameters, change Element Base from Internal to Shared, as shown in Figure 3.57.

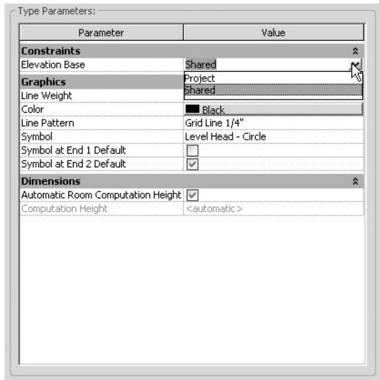


FIGURE 3.57 Changing the element base in the level marker will allow you to "move" the level to the actual datum elevation.

Being able to create and manipulate levels is crucial to how well you do with Revit Structure. Fortunately, this process is basically simple. Remember, though, it is through simplicity that errors can be produced as well. Take nothing for granted, and keep an eye on your model in this stage of the game. The choices you make here, right or wrong, will follow you like a shadow for the life of the model.

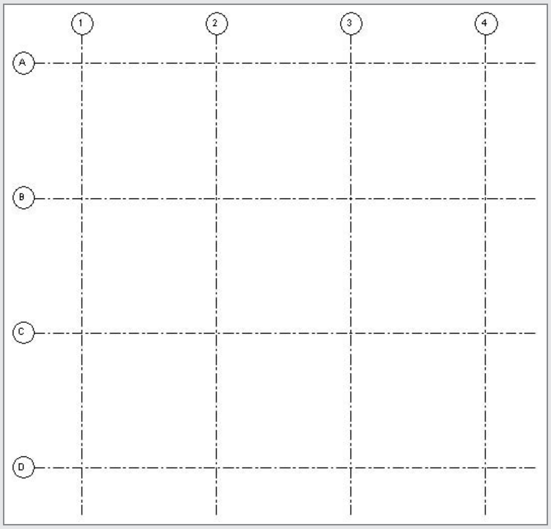
EXERCISE: GRIDS AND LEVELS

This exercise will guide you through the creation of a structural grid as well as addition of elevations. We'll begin using a blank model.

The first set of steps will run through adding a structural grid.

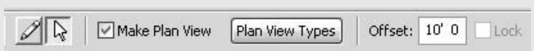
1. Open Revit Structure, and start a new default model.
2. On the Basics tab of the Design bar, click the Grid button.
3. Select a point in the Level 2 plan view window, and then click a point about 75´ below the first point.

4. If you are not still in the Grid command, click the Grid button.
5. On the Options bar, click the Pick button, and type **25´–0˝** in the Offset box. Offset a new grid to the right of the first grid.
6. Repeat the procedure three times. You should be looking at grids 1 through 4.
7. Start the Grid command again, and click the Draw button on the Options bar. This time draw a horizontal grid across the top of the vertical grids.
8. Rename the grid head to **A**.
9. Start the Grid command again, and click the Pick Lines button. Set the Offset to 25´-0˝. Offset grid A down three times at a 25´-0˝ increment. You should have column lines A through D.
10. Select the grids individually and manipulate them.



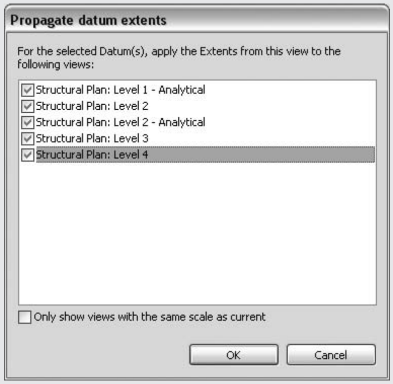
The next set of steps will guide you through creating levels.

1. Go to the South elevation.
2. On the Basics tab on the Design bar, click the Level button.
3. On the Options bar, click the Pick button, and change the offset to 10´-0˝.



4. Offset Level 2 up. You should now have Level 3.
5. Offset Level 3 up. You should now have Level 4. Notice the Project Browser has the new plan view for each of the new levels.
6. Open the Level 4 plan view. Notice there are no grids on this plan view.
7. Go back to Level 1 plan view.
8. Select all of the grids.
9. On the Options bar, click the Propagate Extents button.

10. In the Propagate Datum Extents dialog box, select all of the plan views.



11. Check the other plan views. Now all plan views have the same grids extending into that view.

The Bottom Line

Import and link CAD data. More than half of your projects are going to start with bringing in an architect's CAD data. In this chapter you learned how to bring a DWG file into your Revit Structure model and manipulate it to conform to your company's standards.

> **Master It** Once you add a DWG to the model, you will find that the DWG does not look the way you would like it to. Name two methods of controlling the visibility of the underlay.

Link Revit Architecture. The power of Revit Structure shines when you can get a Revit model from the architect. You can link that model and perform a Copy/Monitor operation to add superior integration unseen in CAD applications.

> **Master It** Although the actual import of the Revit Architecture model is quite simple, you can copy specific objects from the Revit link and keep a live connection telling you if anything changes when the model reloads. How does Copy/Monitor work?

Create structural grids. Revit Structure allows you to create grids "stick by stick." This freedom is crucial to being able to easily model any building shape needed.

> **Master It** In this chapter you learned how to create a grid. Once the gridlines are in place, you have to make further adjustments. Explain how to do so.

Create levels. One of the most compelling aspects of Revit Structure is its ability to contain the entire model in a single file. The ability to create levels and generate floor plan views that are linked to them is a huge part of this functionality.

> **Master It** As mentioned earlier, levels and plan views are connected. How does Revit Structure determine which level belongs to which plan view? What do you do if you need a new plan view based on an existing level?

Part 2

Developing Your Structural Model

Chapter 4

Structural Columns

Structural columns, which are different than just columns (which are called architectural columns in Revit Structure), are what help support almost every building. They are a major part of the skeleton structure that other structure elements like beams and slabs connect to. For this reason, they are probably going to be one of the first or second components that you will model in your project and will be the primary focus in this chapter.

Columns can take on many different shapes and sizes, as well as different construction materials. Depending on the type of material they consist of—wood, steel, precast, concrete—they demand different connection requirements, annotations for location, and documenting of sizes and reinforcing (if any). You should understand the various behaviors each scenario will bring.

Columns are typically defined as a vertical structural member; in today's world, they can become sloped, tapered, skewed, bent, and in some cases spiraled. Yes, you name it, eventually a column will probably have to take on that form. Buildings are no longer square in footprint and straight up vertical in height.

Even though columns are a major part of the building process, the basic structural column is simple to model and work with when using Revit Structure. Those that are not so basic in size and shape or vertical in height can be created and placed in a similar method, but might not take on all of the functionality of a simple column.

In this chapter you will learn to:

- Work with the basic structural column family template
- Place structural columns in your project
- Attach structural columns to other structural components
- Employ the methods of placing sloped columns
- Document your model with the Graphical Column Schedule

Getting to Know the Column Families

Revit Structure has two column template families to help you create the behavior of columns you place in your project: `column.rft` and `Structural Column.rft`. Both templates have Upper Level and Lower Level constraints, as shown in Figure 4.1, which indicate where the top and bottom column should be locked. We'll discuss the use of these levels in the section "Adding Structural Columns to Your Project." Because they are each set to a different category, they will present completely different behavior when they are loaded into your project. So when you are working with Revit Structure, make sure that you are using structural columns and not just columns. There is a dramatic difference, which will limit your capabilities if the wrong ones are used.

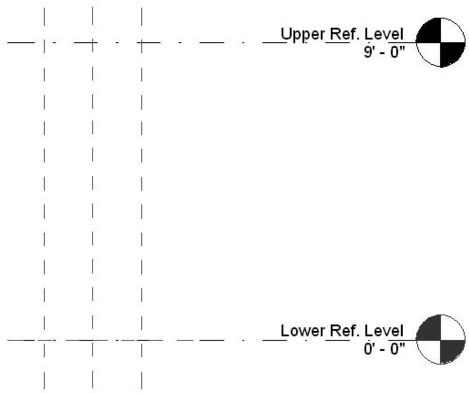


FIGURE 4.1
Upper Level and Lower Level constraints in a column family

A third template is called `Generic Model.rft`. This template does not have the Upper Level parameter, which means you do not have to place it with a Top Level reference. As long as you set the category to Structural Column, it will be available to choose from within the Structural Column list and continue to behave like a column—it just will not be locked to a top level. The reference level in the family will reference to a level of your choice in the project. This level can be used for uniquely shaped and placed columns that are specific to your project.

Figure 4.2 shows a unique structural column that was created with the Generic Model template. This column had two ring beams with varying heights throughout the structure. The top ring supported the roof, and the bottom ring supported the ceiling. Round skylights sat at the top of each column. This unique structural column family obviously does not work with Analysis, but as you can see, it proves to be valuable from a visualization standpoint as well as for documenting the model.

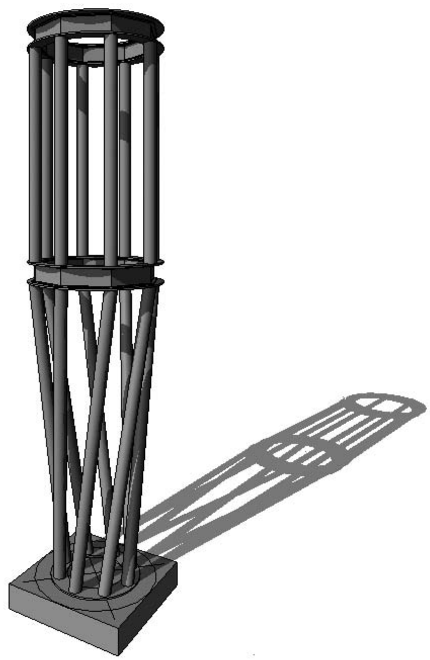


FIGURE 4.2
A structural cluster column created from a Generic Model template

Courtesy of Ericksen Roed & Associates

Getting to know the various column-related categories that are a part of Revit Structure as well as their behavior will allow you to better control how they display in the model and in your documentation. Different templates are available that have certain settings already built into them. You should also understand the various methods and reasons for loading a new column type into your project or duplicating existing types to change their properties.

Columns

You can place columns (architectural columns in Revit Structure) by selecting Modeling ➢ Columns, using the Architectural tab in the Design bar, or dragging the column family type from the Project Browser and dropping it into your project. You probably will find that you will not use these types of columns too much when working on your structure-only projects. When you place an architectural column, the Options bar will display the information shown in Figure 4.3 and will make a minimum of placement methods available, which is quite different from when you place a structural column. For instance, you can only place these columns one at a time, and by default, they will be placed unconnected going up from the level you choose to reference them to. Their behavior will be somewhat identical to how they would behave inside Revit Architecture.

FIGURE 4.3
The Options bar when you place an architectural column

Architectural columns can have the same appearance as a structural column, so from the surface it can be hard to tell which category of column they are set to. Figure 4.4 shows examples of families that have already been created and are available to use as part of the Revit Structure installation.

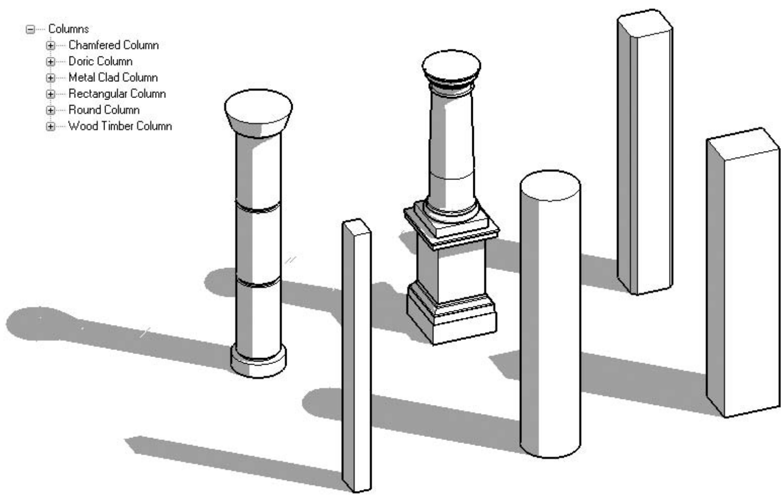
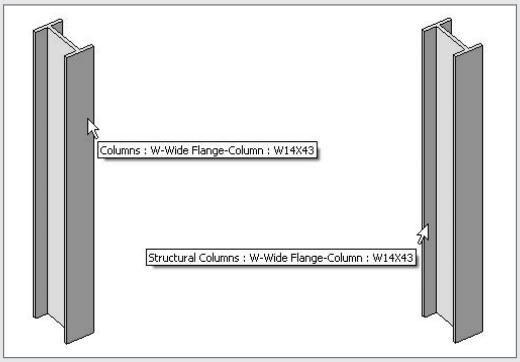


FIGURE 4.4
Architectural column shapes can look just like a structural column.

Another thing to make note of about an architectural column is that structural elements do not recognize them—which means that structural elements do not attach themselves to architectural columns, the columns do not have any analytical information or symbolic display, and the columns will not show up in a Graphical Column Schedule. Once you place an architectural column, you cannot easily swap it out with a structural column, so be sure to make users aware of these differences. Trying to swap the category out inside the family after it has already been placed inside your project may confuse Revit Structure and create errors when you load the family back into your project.

Something Just Isn't Right

If you are noticing strange behavior with your columns or you just can't see them in some of your views, you might want to verify if they are architectural or structural columns. Someone may have accidentally placed an architectural column, copied it from a linked model, or had the mapping information wrong during a copy and monitor exercise. As you can see in the following graphic, you can easily verify which category the column is set to by hovering your mouse cursor over the column and reading the pop-up notification that displays. The first set of words indicates the category name of the object.



If your columns are not displaying, you might want to check the visibility settings of the view to see if the Columns category is checked to display. By default, Revit Structure will have this category set to not display in some views.

Since the architectural column is meant for nonstructural purposes, its built-in family parameters (shown in Figure 4.5) are limited and therefore much different from what you will see when you toggle to the Structural Columns category. These parameters are built into the Family template, which helps Revit Structure understand and control their behavior depending on the family category they are a part of. You can continue to add your own parameters to build additional intelligence into them.

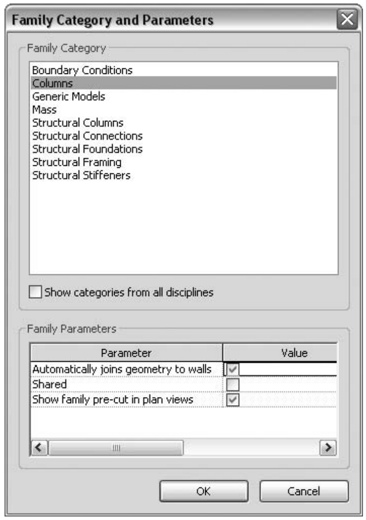


FIGURE 4.5
The Family Category and Parameters dialog box for the Columns category

Structural Columns

Structural columns are the columns that you should be using while modeling in your project. Therefore, the remainder of this chapter will explore the placement and behavior of the structural column. Before starting to place columns, you should have a good understanding of their properties and how they behave when placed in your project.

Structural columns are similar to architectural columns in the way that they look, but they have specific properties that they automatically take on depending on their configuration and industry standards. They also have an analytical representation attached to them, which can be exported and used in other analysis design software.

Unlike walls, slabs, and roofs (which are *system* families), columns are considered an *external* family. This means that they can be created outside your Revit Structure project as independent RFA (Revit family) files and loaded into your project. They also can be created as an in-place family, which you can create directly inside your project. Since they are an external family, you have the freedom to create just about any shape that you want to. Figure 4.6 shows examples of structural column families that have already been created and that are available you to use as part of the Revit Structure installation. These families allow you to go a long way when it comes to modeling, but you will find that for some projects you will have to either modify existing ones slightly or create your own that look totally different.

As you can see in Figure 4.7, the Structural Columns category has a few built-in parameters that allow it to behave differently depending on these parameters' settings inside the family. Setting the Structural material type will add or remove additional parameters that pertain specifically to the material being used. In some cases, you might find that you need to have families that have the same geometry in them, but these options are set so the geometry in each family displays a certain way when placed in your project.

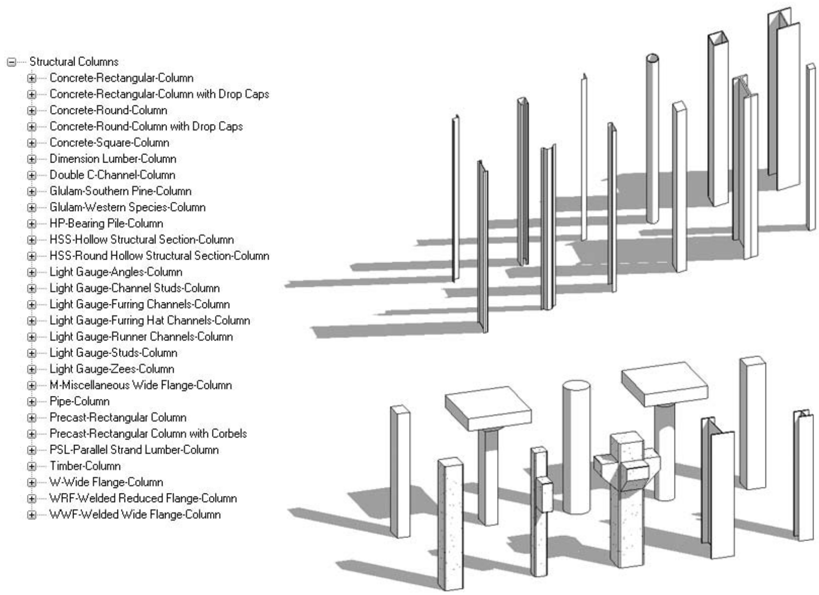


FIGURE 4.6
Revit Structure comes with a large library of structural column families.

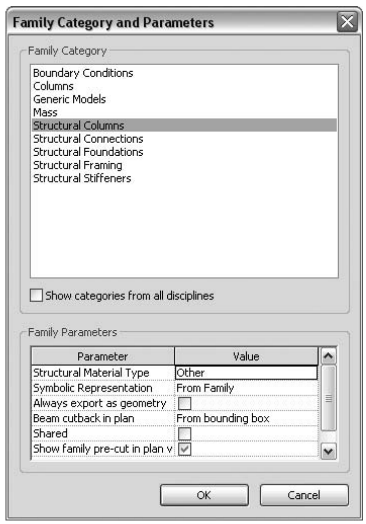


FIGURE 4.7
The Family Category and Parameters dialog box for the Structural Columns category

STRUCTURAL MATERIAL TYPE

When creating your own structural column, make sure that you set the Structural material type inside the family to the proper material that it will take on. You can select from five different material types: Other, Steel, Concrete, Precast Concrete, and Wood. This material property is different from the material that you assign to the family to give it surface and cut patterns. This is

what tells Revit Structure how the family behaves as it interacts with other structural elements and as it displays in certain views. For instance, a concrete beam that frames into a concrete column behaves differently than a steel beam that frames into a steel column or a steel beam that frames into a concrete column. Figure 4.8 shows the automatic behavior that is derived from this setting.

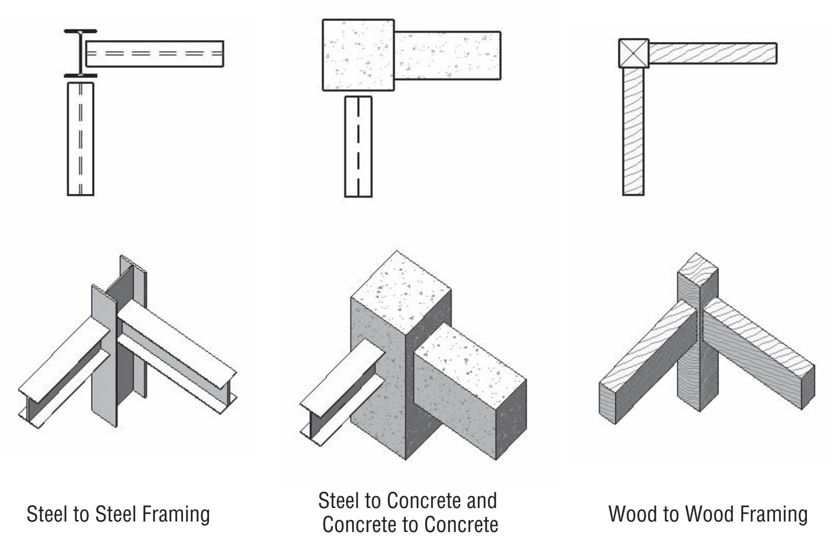


Figure 4.8
The Structural material type setting within a structural column family helps determine how other structural elements attach to them.

The Structural material type helps determine if the two elements should join their materials together with a construction joint or place the steel beam with a setback dimension. Each material behaves differently based on common industry conditions when it interacts with materials that are the same and materials that are different. The Graphical Column Schedule will also use this setting, which allows you to schedule only columns of certain material type. This makes this option one of the most important and the first one you should set when creating your column families.

No, It Really Is Not the Right Material

When you are working with concrete, you might want to join two separate elements together to remove a construction joint. Let's say you need to join a concrete column to a concrete slab and part of the column must be poured with the slab. To join these monolithically, both elements need to have the same material assigned to them. You keep checking the materials, and both the column and the slab have the same concrete material assigned to them. You have toggled between all three different Detail Level settings, and you still cannot get the construction joint to be removed. One last thing to try would be to edit the family and check the Structural material type under Settings ➢ Family Category and Parameters. Chances are that option is set to something other than Concrete.

Symbolic Representation

This setting only affects the family when your views are set to a coarse detail level. If your column family is going to be steel or wood material, you will probably want to have this option set to From Project Settings. That way, Revit Structure will automatically place a single symbolic line at the center of the geometry to represent the column in a plan, section, or 3D view when set to a coarse detail level. The display of this symbolic line can be controlled by the subcategory Stick Symbols under the Structural Columns category. It also controls the symbols that are displayed for the Top and Bottom connection types that are set in the Structural Settings dialog box for the project and an instance parameter of a column. The display of these symbols can be controlled by the Connection Symbols category under the Annotation Categories tab in the Object Styles dialog box. For most companies, this will be the industry standard for showing steel or wood at reduced scales, which would also be shown in Revit Structure's coarse detail level view.

When using concrete or precast concrete, you would typically show the true double-line representation of the shape in all detail levels. For this reason, set the Symbolic Representation option for these families to By Family. This tells Revit Structure to not automatically place symbolic line work and symbols for those families.

Beam Cutback in Plan

This is another setting that only affects the family when your views are set to a coarse detail level. Beam Cutback in Plan refers to the stick symbol Symbolic Line, which Revit Structure automatically generates for a beam family when its Symbolic Representation option is set to From Project Settings. If set to From Bounding Box, the symbolic line will be cut back from the bounding box of the column; if set to From Geometry, the symbolic line will be cut back from the geometry of the column. You are probably wondering, "What is a bounding box?" A *bounding box* is an invisible box that Revit Structure will place to the extents of the geometry inside the family. This concept is best shown with a sloped column, as in Figure 4.9. Revit Structure uses this box to help it make decisions during its automated process. In a beam framing into a column scenario, the Symbolic Cutback Distance setting in the Structural Settings dialog box will start its offset dimension from the bounding box or the geometry of the column, depending on the settings in the Column family.

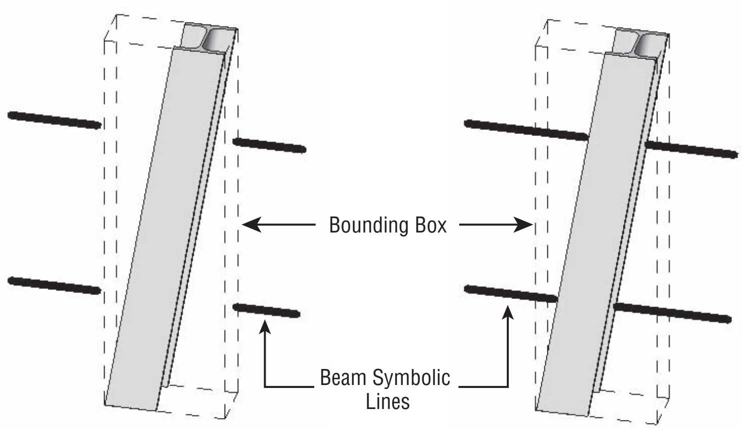
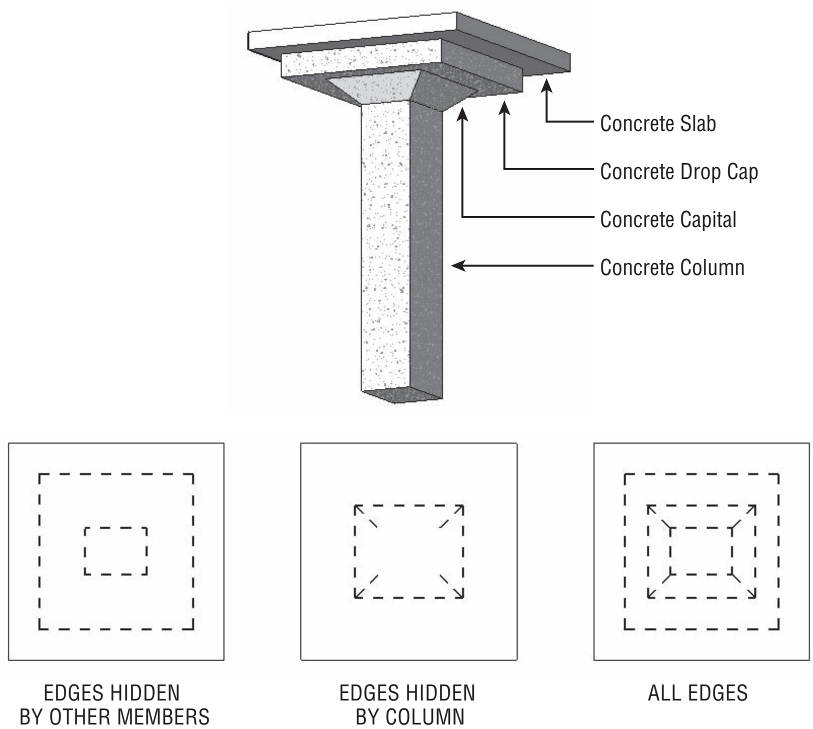


Figure 4.9
For the column on the left, Beam Cutback in Plan is set to From Bounding Box; for the column on the right, Beam Cutback in Plan is set to From Geometry.

Display in Hidden Views

When the Structural material type is set to Concrete or Precast Concrete, this parameter becomes available to help control the visibility of hidden lines that should or should not display in your concrete column families. There are three settings for this parameter: Edges Hidden by Column, Edges Hidden by Other Members, and All Edges. Figure 4.10 shows the various displays that Revit Structure will produce depending on what your family is set to.

Figure 4.10
Setting the Display in Hidden Views option in the Structural Column family produces various plan displays.



In some cases, you may have to have an exact duplicate of your family with a different family name. The only difference between the two families may be the Display in Hidden Views setting. Sometimes creating duplicate families with different settings will have to be done so your column with special conditions display properly once inside your project.

Family Loading and Duplication

Since structural column families are external families, you will need to import them into your project in order to use them. Once the families are inside your project, you will be able to duplicate them to create different sizes. Some structural column families will be created with a type catalog file that is part of the family. This type catalog file allows you to define a list of preset types that will allow you to lock in the settings for each type and display them in a list to choose from when importing. More can be found on the use of Type catalogs by searching the Help index in Revit Structure.

To load a structural column family or any other external family, choose File ➢ Load from Library ➢ Load Family. Browse to the location of your structural column family and select it to open it. Several families can be loaded at once, similar to Windows' standard multiple file selection if you hold down your Shift or Ctrl key while selecting the files. If a family is using a type catalog, Revit Structure will display a list of preset types, as shown in Figure 4.11, for you to choose from. In this list you can load only the types that you need.

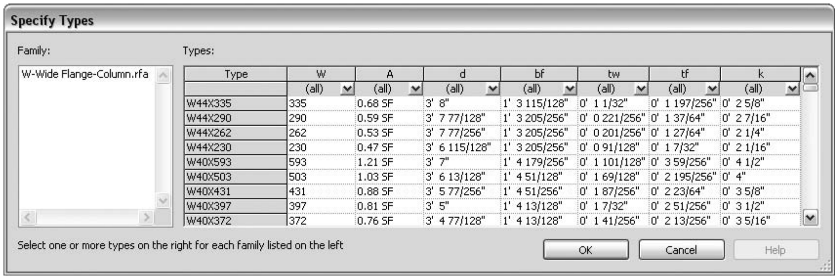


FIGURE 4.11
Using a type catalog for a family to load preset types into your project

Type catalogs are typically used when the structural shape properties are pretty much static for their type. The properties of steel, light gauge steel, and wood shapes are usually pulled right from a product catalog or are industry standards. Structural elements that are concrete or precast concrete can come in just about any shape you could imagine, so putting them in a type catalog would be exhaustive.

Real World Scenario

REMEMBER TO CHANGE THOSE PARAMETER VALUES

When duplicating types inside your project, it is important to remember that just duplicating a structural column type with a name of W8X31 and giving it a name of W12X45 does not automatically change the parameter values that give the type its physical properties. All parameter values need to be changed after duplication to completely define it as a new type.

One example comes to mind: we had a user who was new to Revit Structure and was modeling a five-level composite steel structure. The template at the time had only one wide flange column loaded in it (a W8X31). As new column sizes were needed, the user continued to duplicate the W8X31. Of course, the duplicates all had names that reflected the shape of the new columns, but all 10 columns had the size properties of a W8X31. Not a good example of a true BIM project…

When working with families that are using type catalogs, you should load in new shapes from the catalog rather than duplicating them. You load new types in the same way you load the family for the first time. Use the Duplicate method for families that do not use type catalogs.

To duplicate a Structural Column #1, perform the following steps:

1. Select a column(s) that you want to duplicate.
2. Open its Element Properties dialog box.
3. Click the Edit/New button in the dialog box.
4. Click the Duplicate button in the Type Properties dialog box.
5. Give the duplicated column a new name.
6. While still in the Type Properties box, make changes to all parameters that differ from the other types.
7. Click OK in all subsequent dialog boxes.

To duplicate a Structural Column #2, perform the following steps. (This method allows you to create new types without touching already modeled elements.)

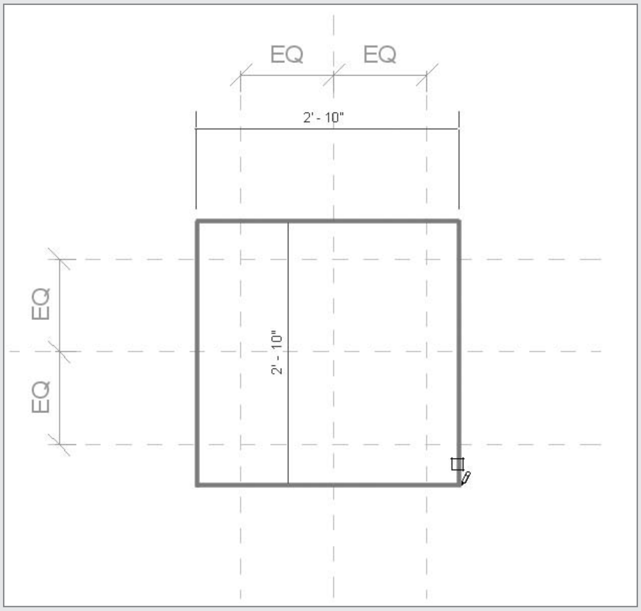
1. Select a column type from the Project Browser in the Structural Columns category.
2. With the column type selected, right-click and select Duplicate.
3. Give the duplicated column a new name.
4. Right-click on the new column type and select Properties.
5. While in the Type Properties dialog box, make changes to all parameters that differ from the other types.
6. Click OK in all subsequent dialog boxes.

Exercise: Creating a Simple Structural Column Family

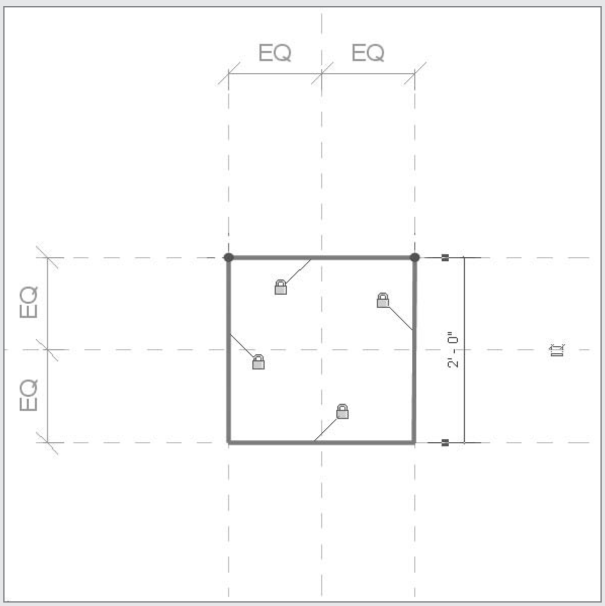
In this exercise, you'll use a simple structural concrete column, one that is already created for you in the Revit Structure installation. Going through the steps will help you grasp the idea and help you create structural columns of much greater complexity.

1. Choose File ➢ New ➢ Family.
2. Browse to `Imperial Templates\Structural Column.rft` and select it to open it.
3. Choose Settings ➢ Families Category and Parameters.
4. Set the Structural Material Type option to Concrete. Click OK to close the dialog box.

5. While in the plan view, click the Family tab on the Design Bar. Locate and click Solid Form and then Solid Extrusion. Select the Rectangle line tool on the Options bar and place the sketch lines outside the reference planes, as shown here:

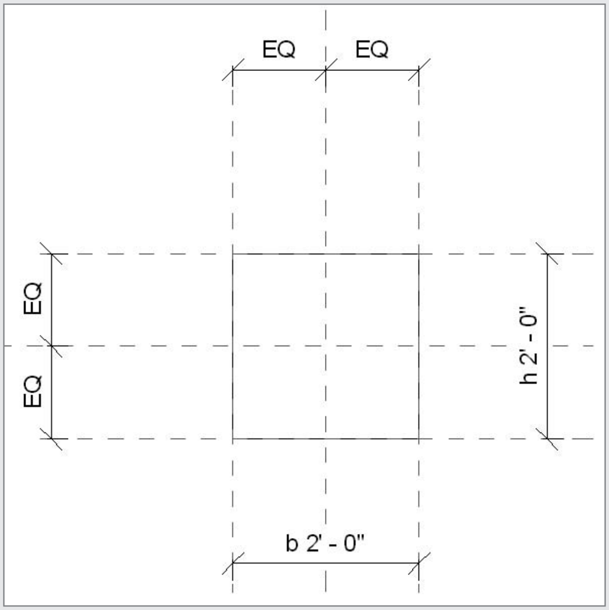


6. Use the Align command on the toolbar to align and lock the sketch lines to the reference planes, as shown here:



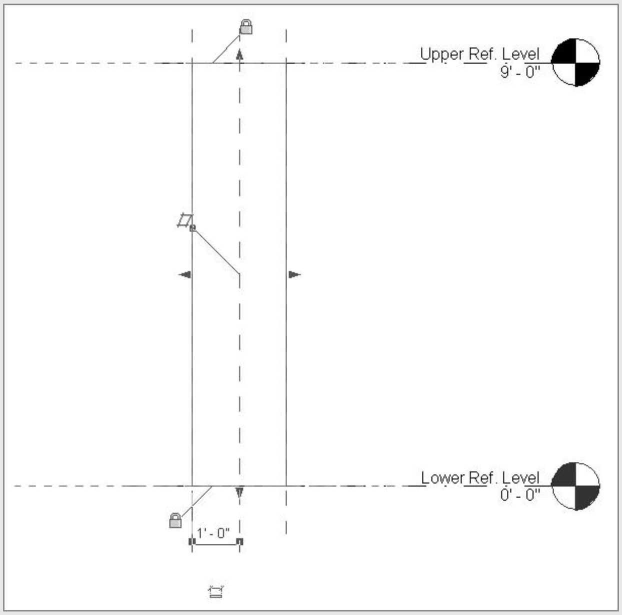
Note that placing the initial sketch lines away from the reference planes and then aligning and locking to them afterward ensures that your sketch lines are properly locked to the reference planes.

7. Click Finish Sketch on the Design bar.
8. Place dimensions to the reference planes for both the width and height of the column.
9. Create a Label parameter, as shown in the following graphic, so the extrusion will flex inside your project. You can easily do this by selecting the dimension(s) and then selecting Add a Parameter from the Label box on the Options bar.



10. In the Parameter Properties dialog box, fill out the appropriate information by giving the parameter a name (here we use **b** and **h**) and group them under Dimensions. Make them a type parameter and click OK.
    Note that when using this method the Type of Parameter value has already been set to Length. This is because a Length parameter is the only type that can be assigned to a dimension.
11. Select the Front Elevation view from the Project Browser.

12. Move the bottom of the column off the Lower Reference level and then align and lock the top and bottom of the column extrusion to the upper and lower levels as shown here:



13. Select the extrusion and open its Element Properties dialog box.
14. In the Element Properties dialog box, select the little rectangular box next to the Material parameter.
15. In the Associate Family Parameter dialog box, click Add Parameter.
16. Give the parameter the name **Material**, make it an instance parameter, and group it under Material and Finishes.
17. Click OK to return to the Element Properties dialog box.
18. Notice the rectangular box next to the Material parameter now has an equal sign in it. Steps 14 and 15 just added a parameter to your family and hardwired it to the extrusion, so when you place the parameter in your project, you are able to assign different materials to each instance of a column that you place. Click OK to close the Element Properties dialog box.
19. Perform a few safety checks by changing the Dimensions values in the Family Types dialog box on the Design bar to make sure that your new family flexes properly.

Adding Structural Columns to Your Project

After learning about the various column family libraries and what makes them tick, you are ready to move on to placing columns in your project. You can use the `Structural Columns.rvt` file for this portion of the chapter. Before placing structural columns, you should have levels and a pretty good portion of your grids generated. The top and bottom of any columns will need to refer to levels that are already generated in the model. If you place columns on grids, you can set them to stay attached to the grids. That way, when a grid moves, the column will move. The grids will also allow modeled columns to display in a Graphical Column Schedule.

The natural tendency when starting a project is to model your columns full height from the top of the foundation to the roof. This makes perfectly good sense in a single-story structure. But you have to stop and ask yourself, "What concrete column is poured in a 100′ lift?" or "Who erects a 100-foot-long steel column?" The preferred method is to place columns as the building will be built.

If you are modeling a concrete post-tensioned or flat plate structure and the columns go floor to floor, you should model the columns floor to floor. A good rule is to stop the columns at the construction joint—which will most likely be the top of the slab for the bottom of the column and the bottom of the next slab above or below the beam for the top of the column. Typically, concrete structures are erected floor by floor so the concrete columns would be placed level to level in Revit Structure.

If you are modeling a steel structure, the columns are more than likely going to be erected to a certain height before they will need to be spliced. This might be necessary to meet Occupational Safety & Health Administration (OSHA) requirements or because of a change in column size. A steel column usually projects above a specific level before it splices, stopping just below a level at a roof or termination of column condition. Regardless of the scenario, you should be modeling columns as they will be built.

Not only does modeling structural columns as they are constructed give you a more accurate BIM model, but when it comes to creating sections and Graphical Column Schedules (GCSs) and performing quantity take-offs, you will spend less time developing your sections, create a useful schedule, and have a much more accurate representation of material amounts.

The biggest part of mastering the placement of structural columns in your project is learning the ways that you can place them in your project and knowing the best times to use those particular methods. Thinking ahead in the modeling process by attaching structural columns to other elements (so they adjust to changes in the model) will help keep element relationships consistent and their parametric behavior as you intended.

Placement

To begin placing your structural columns, select Structural Columns from the Modeling tab on the Design bar, choose Modeling ➤ Structural ➤ Structural Column, or drag the family type from the Families section of the Project Browser and drop it into your project. A fourth method is to right-click on an existing column and select Create Similar from the context menu. The Options bar offers several methods for placing structural columns (Figure 4.12).

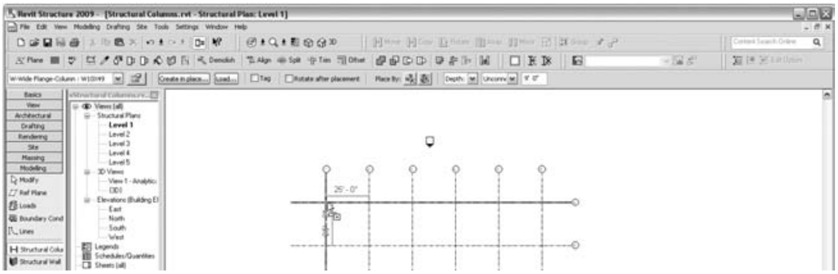


Figure 4.12
The Options bar gives you several options for placing a structural column.

Reading from left to right, the Options bar allows you to do the following:

- From the Type Selector pull-down, select the type of structural column you want to place.
- Before placing a column, you can select the properties of the currently selected column to make changes to its properties or duplicate it to make a new column type.
- If the column you are creating is unique to the project, you can select Create in Place. Doing so automatically locks the category of the In Place family to the Structural Column category.
- If you want the columns to be tagged in the current view directly after placement, select the Tag check box.
- Selecting the Rotate After Placement option automatically puts you in the Rotate command after placement, with the center of the column serving as the center of rotation.
- Two additional placement methods are grid intersection and architectural columns. These are both explained further below in “Using the Grid Intersection Option” and Using the Architectural Column Method.”
- In the Height/Depth area, you can choose to place the column with a depth (going down) or with a height (going up) with a reference to the current level you are placing it in.
- To the right of the Height/Depth area is the Constraint list, where you can set the constraint of the top or bottom of the column or set it to be unconnected.
- If using an unconnected height, you can give a depth or height of the column from your current level.

After columns are placed, take a look at the properties of a few of them. You will find that their instance parameters will vary depending on the structural material type assigned to them in their family as well as how they are placed in the model. Figure 4.13 shows that a steel column will have additional graphic parameters for the display of symbolic symbols and concrete columns have additional structural parameters for the concrete cover.

Every structural column will have a base and a top level that the top and bottom of the column refer to. From each of those references, you can specify a top or bottom offset from the specified reference level. If you have an interior column sitting on a footing that is 8″ below the Level 1 slab on a grade, the Base Level option of the column would be set to Level 1 with a Base Offset dimension of –0′-8″. The same column that goes up and splices 4′-6″ above Level 3 would have its Top Level option set to Level 3 with a Top Offset setting of 4′-6″.

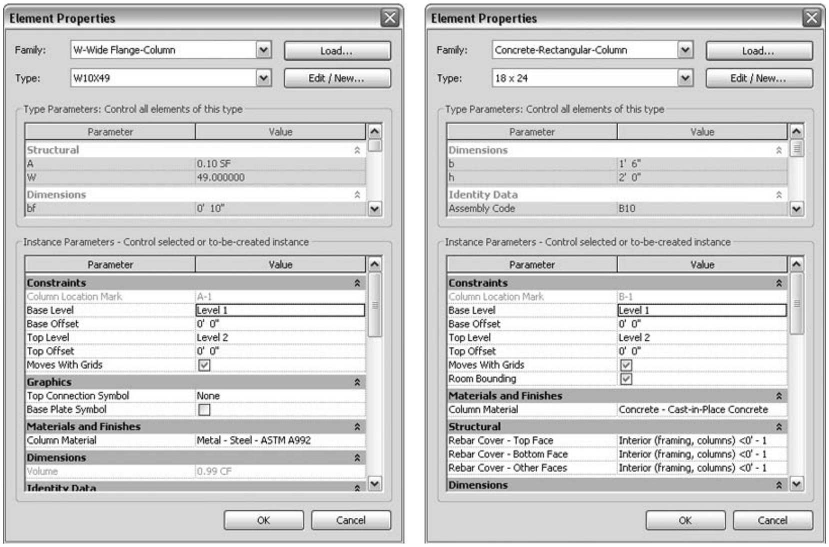


FIGURE 4.13
Structural column properties vary depending on their structural material.

When you're placing structural columns in Revit Structure, several tools are available that allow you to place them quickly as well as ensure that they are placed properly. Using the Single Pick option will place a single column one by one and also control the rotation of each specific placement. Using the grid intersection or architectural column placement methods allow you to place several columns at once while using other elements for their placement location.

USING THE SINGLE PICK OPTION

The Single Pick option is the initial state that Revit Structure puts you in after you select one of the structural column placement commands. This allows you to place columns one at a time and easily adjust settings between each placement.

To use the Single Pick option, follow these steps:

1. Columns will be placed down by default, so activate an upper-level plan view.
2. Select the Basics or Modeling tab on the Design bar and choose Structural Column.
3. Select the type of column you want to place.
4. Observe the Options bar and make any necessary settings. Click the Properties button and set the Top Level and Base Level values as well as any offsets that are required.
5. Start placing columns one by one as they snap to the intersection of gridlines.
6. You can rotate columns while placing them by pressing the spacebar. Each tap of the spacebar rotates the column 90 degrees. If on gridlines, the column will snap perpendicular to them and use an angle degree that's half the grid intersection angle.
7. You can tag each column as you place it by selecting or deselecting the Tag box in the Options bar.

Using this method is useful for placing columns that are not on grids or that require a more specific location or rotation. Even after columns are placed, you can use the spacebar to rotate them by selecting one or two columns and pressing the spacebar.

If you use this method for grid intersection placement, take care to ensure that you are correctly placing the column at the intersection point rather than another unexpected intersection point. Keep an eye on the status bar located in the lower-left area of your Revit Structure session dialog box to verify where the column is being placed. Setting the visibility properties of a view to only show grids and columns may help you select the correct intersection point.

Using the Grid Intersection Option

The Grid Intersection option allows you to select groups of grids for placement. Revit Structure will place a column on all grid intersections that you select. This can be a quick method for getting columns into your project. Even if a column is not supposed to be on a particular grid intersection, it is still fine to place the column on it and remove it afterward.

Ask yourself which way you can do it faster. Should you place 50 columns by using the Single Pick option, or should you place 55 columns with the grid intersection method and erase five of them afterward? Figure 4.14 shows that you can have 30 columns placed onto your gridlines in a matter of seconds.

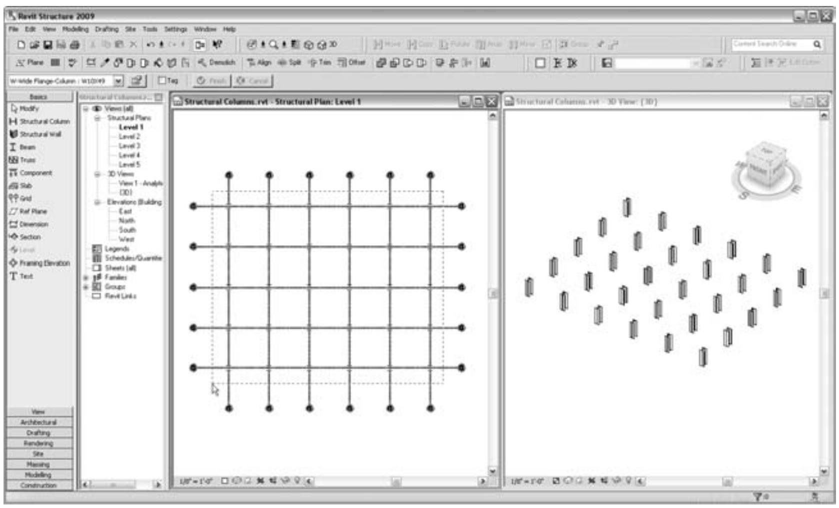


Figure 4.14 Placing columns by grid Intersection will quickly and accurately place them onto gridlines.

For grid intersection placement, perform these steps:

1. Columns will be placed down by default, so activate an upper-level plan view.
2. Select the Basics or Modeling tab on the Design bar and choose Structural Column.
3. Select the type of column you want to place.
4. Observe the Options bar and make any necessary settings.

5. Select the Grid Intersection Placement option on the Options bar.
6. While in this mode, your Options bar will refresh to display the option to Tag the column(s) after placement, Cancel out of the tool or Finish to accept your Grid Selection.
7. Select all grids with a right-to-left crossing window. You should see columns display at the center of all selected grid intersections.

Select grids by clicking the gridlines and holding down the Ctrl key to add to your selection, or by holding down the Shift key to subtract from your selection. You should see columns appear and disappear at the center of grid intersections as you add and subtract grids from your selection.

8. Once all required grids are selected, be sure to click the Finish button on the Options bar to accept your column placement. Any other action will remove your placements.
9. You can now delete any unwanted columns that are on grid intersections and do any fine-tuning of their rotation.

Using this method will help ensure that all columns are accurately placed at the exact intersections of the grids. This will also ensure that the columns will properly be attached to the grids so that when the grids move, the columns will move with them. When columns are properly placed at grid intersections, they will continue to display in the Graphical Column Schedule (which we discuss later in this chapter).

Using the Architectural Columns Method

If you have architectural columns modeled in your project, you can use them to place your structural columns. As shown in Figure 4.15, Revit Structure will place a column at the center of all architectural columns that you select. If this method of placement fits in with your modeling workflow, this can be a quick method of getting columns into your project.

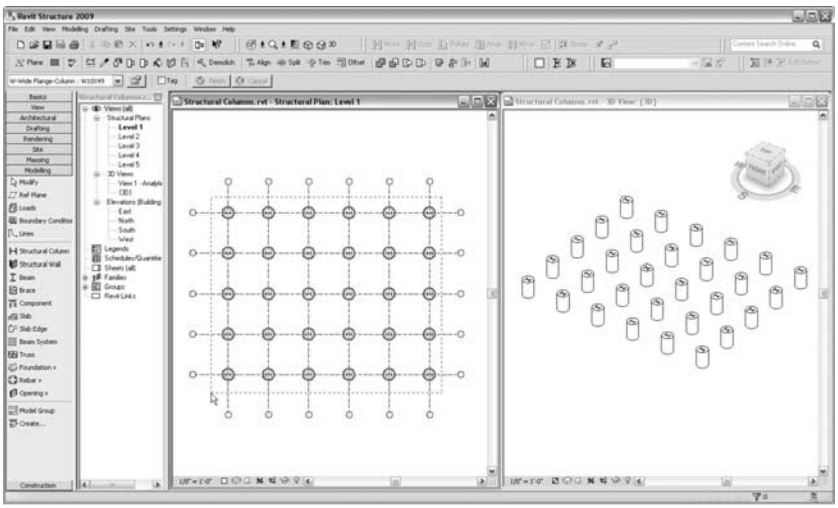


Figure 4.15
Revit Structure will place a column at the center of all architectural columns that you select.

For architectural column placement, perform the following steps:

1. Columns will be placed down by default, so activate an upper-level plan view.
2. Select the Basics or Modeling tab on the Design bar and choose Structural Column.
3. Select the type of column you want to place.
4. Observe the Options bar and make any necessary settings.
5. Select the Architectural Columns Placement option from the Options bar.
6. While in this mode, your Options bar will refresh to display the option Tag After Placement, Cancel or Finish Your Grid Selection.
7. Select all architectural columns with a right-to-left crossing window. You should see columns display at the center of all selected architectural columns.

   Select architectural columns by clicking them and holding down the Ctrl key to add to your selection, or by holding down the Shift key to subtract from your selection. You should see columns appear and disappear at the center of each architectural column as you add and subtract them from your selection.

8. Once grids are selected, be sure to click the Finish button in the Options bar to accept your column placement.

If you are using this method to place your columns and the architectural columns that you're selecting are from an outside client, make sure that architecturally the columns are accurately placed. Usually an architectural column will be in the form of a column surround, so when you place the structural column you place it in the surround. Communicating the behavior of the columns to those who created them will help achieve an accurate placement.

Some companies may use the structural linked model to display the structural column in their model, so this can be an excellent way to collaborate and keep your documents coordinated.

Copying Columns to Other Levels

Once you have columns placed in your project for one of your levels, another time-saver you can take advantage of is to copy them to other levels as needed. Each project will be a little different, so you will have to use your judgment on which tools and methods are best to use.

The basic procedure goes like this:

1. Select the column(s) that will be copied up to other levels.

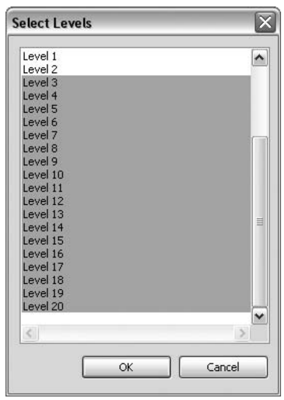
   Switching to a 3D view to perform your selection and then using the Filter Selection tool to select only your columns can speed up your selection process. The 3D view allows you to orient to an elevation view that lets you select through the model with no clipping plane.

2. Choose Edit ➢ Copy to Clipboard or press Ctrl+C on your keyboard.
3. Choose Edit ➢ Paste Aligned. Paste Aligned will provide several options to choose from, depending on which view you are in when you issue the command. Your options are:
    - Current View
    - Same Place
    - Pick Level Graphics
    - Select Levels by Name
    - Select View by Name

    After you select a Paste Aligned option, the columns will be placed into your model depending on which method you select.
4. While columns are still selected, right-click and choose Properties from the context menu.
5. Make any parameter adjustments that are needed for the final location of your columns.

Using Paste Aligned ➢ Select Levels by Name allows you to select several levels at once. If you had a 20-story concrete structure and you had the columns already placed on the first level, you could select and copy all of those columns to the clipboard. Figure 4.16 shows how you could then paste them to each of the other levels by selecting all of them. This would place an individual column directly above the lower columns at each level. In the scenario shown, the columns that were copied were referenced from Level 1 to Level 2. Selecting Level 2 would result in placing columns over the top of the ones that you copied; selecting Level 1 would result in placing the top of the column referenced to Level 1 with the bottom projecting below.

FIGURE 4.16
Selecting multiple levels to paste columns to other levels



Exercise: Place Structural Columns for a Steel Structure

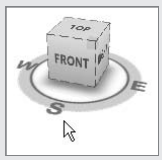
For this exercise, you can use the `Structural Columns-STL.rvt` file. You will go through the steps of placing columns into your project first by placing the first lift . You'll then use the Copy to Clipboard method to place the upper levels. Columns will span two supported levels before creating a splice connection.

1. Open the `Structural Columns-STL.rvt` model.
2. Open the Level 3 View (if necessary).

   Note that the first lift of columns will go from Level 1 to Level 3.
3. Select the Basics or Modeling tab on the Design bar and choose Structural Column.
4. Select W10X49 from the Type Selector pull-down.
5. Set the Column constraint option to Depth and the bottom constraint to Level 1.

6. For the Place By option, choose Grid Intersections.

7. Select all grids from upper right to lower left. Temporary columns will display.
8. Click Finish on the Options bar.
9. Delete the columns on grids A-4, A-5, A-6, B-4, B-5, and B-6.
10. Open into a 3D view and orient the view so that you're looking south by choosing View ➢ Orient ➢ South or by selecting the "S" on the ViewCube, shown here:

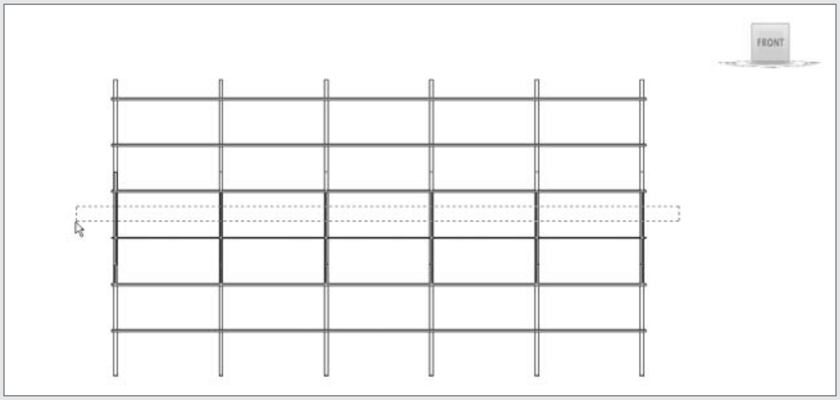


    When you select columns in the next steps, it appears that you are only selecting six columns when creating a right-to-left crossing window. Since a 3D view does not have a clipping plane, you actually are selecting all columns.
11. Select all columns that you previously placed with a right-to-left crossing window, right-click, and select Element Properties.
12. Change the Top Offset value to 4´-6˝ and close the Element Properties dialog box.
13. While columns are still selected, choose Edit ➢ Copy to Clipboard.

14. Choose Edit ➢ Paste Aligned ➢ Select Levels by Name and select Level 5 and Level 7.

    The levels were selected for the tops of the columns because Revit Structure will place them with depth when they are pasted.

15. Select the second lift of columns that go from Level 3 to Level 5 with a right-to-left crossing window (as shown here), right-click, and select Element Properties.



16. Change the Base Offset value to 4´-6˝ and close the Element Properties dialog box.
17. Select the third lift of columns that go from Level 5 to Level 7 with a right-to-left crossing window, right-click, and select Element Properties.
18. Change the Base Offset value to 4´-6˝ and the Top Offset value to –0´-5˝ (the depth of the slab) and close the Element Properties dialog box.
19. Check that all columns are placed correctly.

Using a top offset for the column that matches the thickness of the slab will not create a relationship between the column and the slab. If the slab changes thickness, you will have to change the Top Offset value accordingly. Another option is to attach the columns to the bottom of the slab. With this method, the top of the column will automatically adjust as the slab thickness changes. To view a completed model of this exercise, see the `Structural Columns-STL_Complete.rvt` file.

Top and Bottom Attachment

Like walls, the top and bottom of columns can be attached to other structural elements such as floors, roofs, foundations, and structural framing members to help maintain their relationship. They can also be attached to reference planes and reference levels. The Options bar will switch to display Attach and Detach buttons, as shown in Figure 4.17, when you select a structural column. When you click the Attach button, the Options bar will switch to display settings for the type of attachment you want to make.

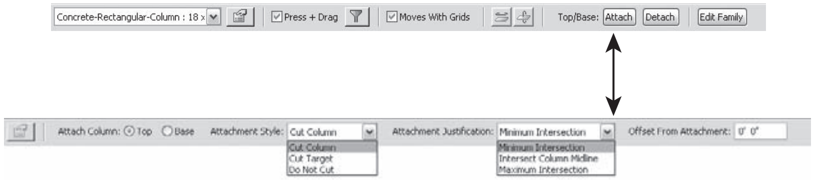


FIGURE 4.17
Options bar attachment

Reading from left to right, the Options bar in Figure 4.17 allows you to do the following:

- Select Top or Base to define which end to apply the attachment to.
- Select the attachment style type.
- Select the attachment justification type.
- Define any offset from the attachment that may be required.

To attach the top of a structural column to a slab:

1. Select the column(s) that will be attached.
2. On the Options bar, click the Attach button.
3. Select Top on the Options bar.
4. Set the Style and Justification method on the Options bar.
5. Select the slab that you will be attaching to.

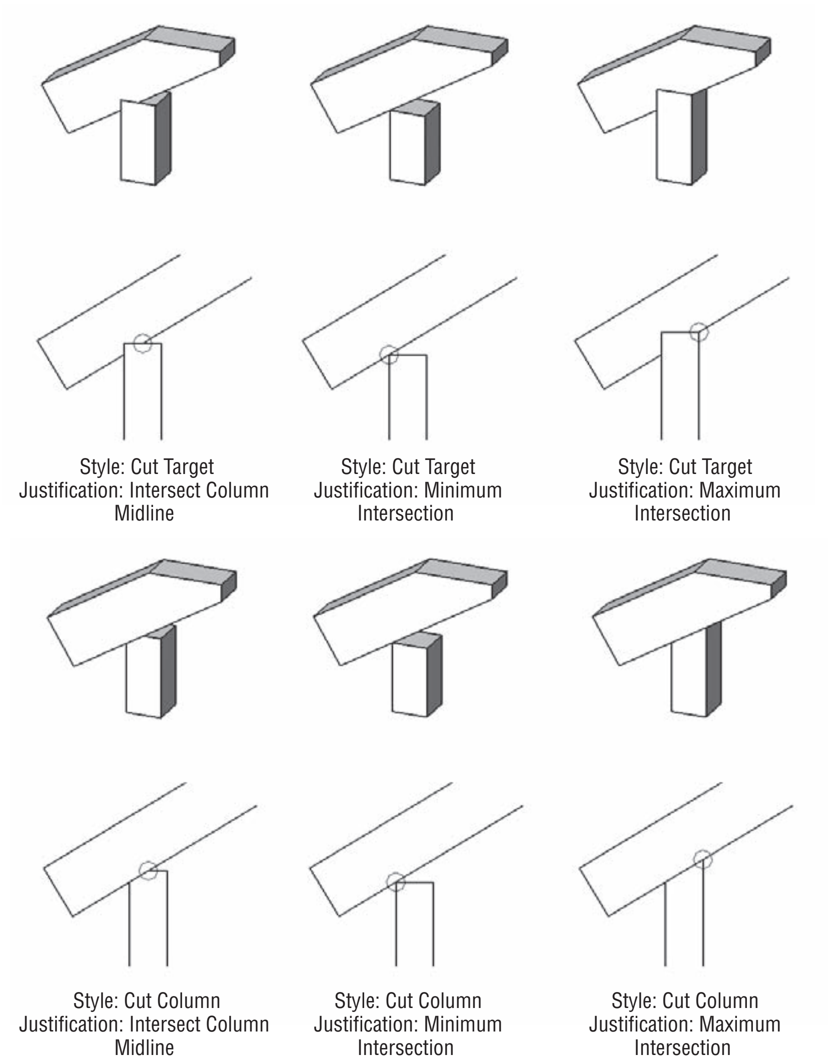
To attach the bottom of a structural column to the top of a beam:

1. Select the column(s) that will be attached.
2. On the Options bar, click the Attach button.
3. Select Base.
4. Set the Style and Justification method.
5. Select the beam that you will be attaching to.

ATTACHMENT STYLE AND JUSTIFICATION

The style and justifications will react differently depending on the type of element and the material are that you are attaching to. Concrete-to-concrete attachments will result in an automatic join in which neither the column nor the target will be cut. A steel column can cut to a steel beam, but a concrete column will not. We encourage you to spend a few minutes playing around with the different style and justification types and see how they react with various material types and elements. Knowing these limits will help you put constraints in your model that will save you time further into your project. Figure 4.18 shows examples of various style and justification combinations.

FIGURE 4.18
Setting the proper attachment style and justification can eliminate tedious detail cleanup.



Revit Structure is capable of dealing with most of the different requirements of how a structural column must perform. The tools provided to place them into your project as well as help maintain their behavior with other elements work in sync with how you need to work with them when you are modeling. In some cases, you will need to go outside the tools that are part of Revit Structure to help show certain column types or shapes that are not yet part of the core program.

Exercise: Place Structural Columns for a Concrete Structure

For this exercise you can use the `Structural Columns-CONC.rvt` file. You will go through the steps of placing columns into your project by placing the first lift, then using the Copy to Clipboard method to place the upper levels. Individual columns will span between each supported level and will be attached to the bottom of the concrete slab.

1. Open the `Structural Columns-CONC.rvt` model.
2. Open the Level 1 View.

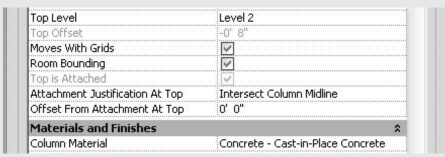
Note: The first lift of columns will go from Level 1 to Level 2.

3. Select the Basics or Modeling tab on the Design bar and select Structural Column.
4. Select Concrete-Rectangular-Column: 18 x 24 from the Type Selector pull-down.
5. Set the Column constraint option to Height and the top constraint to Level 2.

6. Select Grid Intersections for the Place By setting.

7. Select all grids, from upper right to lower left. Temporary columns will display.
8. Click Finish on the Options bar.
9. Delete the columns on grids A-4, A-5, A-6, B-4, B-5, and B-6.
10. Display a 3D view.
11. Select all columns with a crossing window. If more than just the columns are selected, use the Filter Selection tool on the Options bar to only select the columns.
    Using the Filter Selection tool will allow you to include only the elements in your window selection that are in the category that you choose to select in the Filter Selection dialog box.
12. Click Attach on the Options bar.
13. In the Options bar shown here, attach the column at the top with a style of Do Not Cut and a justification of Column Midline.

14. Select the bottom of the Level 2 slab.
15. While the columns are still selected, right-click and select Element Properties.
16. Check the Top Attachment parameters, shown here, and close the Element Properties dialog box.



17. Select all columns.

18. Choose Edit ➢ Copy to Clipboard.

19. Choose Edit ➢ Paste Aligned ➢ Select Levels by Name and select Levels 3 through 13.

If the floor-to-floor heights are the same as the columns you are copying from, then the columns will be placed properly. If not, the columns will be placed with incorrect top or base offset values. Revit Structure will not automatically increase or decrease the height of the column to align with the new levels the column is being associated to.

For example, if the original columns referenced to Level 2 and Level 3 with a height between them of 10´-0˝ were pasted to Level 5 and Level 6 with a height between them of 11´-6˝, the pasted columns would be referenced to Level 5 and Level 6 with a Base Offset value of 1´-6˝. You would see a gap between the slab and the bottom of the column. These columns would then need to have their Base Offset value reset to 0´-0˝.

20. Verify that all columns are placed correctly.

The columns that originally had their top attached to Level 2 maintained their top attachment to each level that they were copied up to. If any of the slabs change thickness or location, the top of the column will automatically stay attached to the bottom of the slab and update the Top Offset value accordingly. To view a completed model of this exercise, open the `Structural Columns-CONC_Complete.rvt` file.

That Isn't All You Are Good For

Structural columns don't always have to be used for columns. They can also be used as piers. Yes, you would think that a pier would be part of the Foundation or Wall category since that is where a pier is usually placed. Sometimes a pier can be part of a wall, and other times it might act as a column. When piers are part of the Structural Column category, they will behave just like a column. They will show up in the Graphical Column Schedule, they will stay attached to grids, and if placed correctly, they will let isolated footings attach to them automatically. When placed in a wall, they will autojoin to the wall.

Another thing structural columns can be used for is hangers. Hangers might not be classified as a column, but they take on the same shapes and have the same characteristics as far as how you would want to place them. A hanger is still going to have a top and bottom elevation to them. You can also attach the top of a hanger to the bottom of a beam; when the beam size changes depth, the hanger will adjust with it.

When using structural columns as a pier or a hanger, you might want to rename the family to something closely related to being a pier or hanger and assign it to a subcategory so you can more selectively control its visibility and how you work with it inside the project.

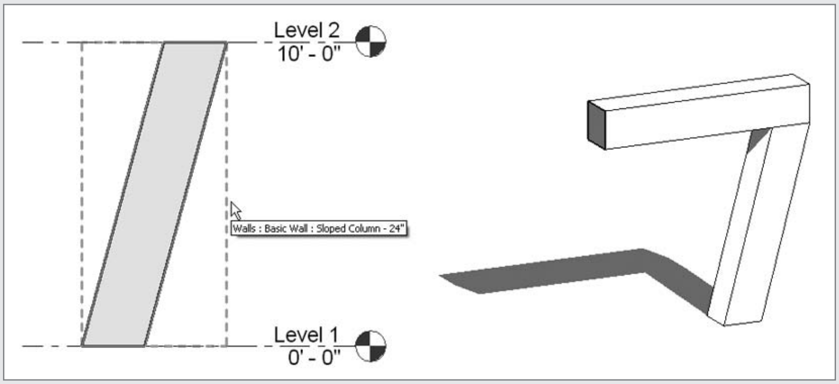
Adding Sloped Columns to your Project

There are some cases, such as with sloped columns, where Revit Structure does not fully deal with the complexity that can be involved when working with their structural makeup, relationship to other structural elements, analytical capabilities, and how they need to be shown for documentation. You have other approaches to deal with such conditions. Depending on the number of sloped columns in your project or how you need them to perform, be selective in

choosing which method is best for you. Since Revit Structure does not have the ability to place these types of columns, other structural elements will not know how to attach to them, views will not know how to display them, and analytical programs will not know how to understand them correctly. Keep these factors in mind when performing a workaround to get sloped columns into your project. Each method has its own pros and cons.

Hmmm, a Wall for a Column

If you are trying to model a concrete column that includes a taper, slope, or other odd configuration, consider using a wall. You can duplicate a wall and assign it a specific name, like *Sloped Column - 24˝*. The wall can be modeled to the extent of the column shape, and then you can edit the profile of the wall by choosing Edit Profile on the Options bar when the wall is selected. When in the edit mode, you can add, remove, or rework sketch lines to produce the shape of the column. Here is an example of a concrete sloped column with a concrete beam running over its top. The sloped column is modeled as a wall with its profile edited to create the shape of the column.



This method will give you a column with the appearance of a sloped concrete column, but it will still have the same properties of a wall and schedule as a wall. You can generate other forms with this method. If you are only looking for appearance, then this might work for you.

Beams set to referenced planes or modeled as bracing can be used to produce a sloped column condition. You can then use the Opening by Face or use a reference plane to cut the geometry to shape the ends. With this method you will find that you can get columns close to how you want them to look in a 3D view but the display in plan will not be as satisfying. You may have to modify the beam family to assign additional symbolic display or add subcategories to give you the ability to turn the display of elements on or off depending on which type of view you are in.

The analytical line that will be placed with this method follows the slope of the beam. This behavior will work better for any analysis programs that are being used. However, within the analysis programs the sloped member will still be detected as a structural framing member and not a column. If you use columns that are part of the Structural Column category, they will continue to show the analytical line as a vertical line rather than follow the slope of the column. Also note that when you're using beams or bracing to simulate a sloped column, they will

continue to be a part of the Structural Framing category and thus will display and schedule as structural framing.

Most sloped columns are modeled as an in-place family; they are fairly specific to the project and do not need a lot of flexibility. If more flexibility is needed, you may want to resort to using an external family, where you can put as much intelligent information and geometry control into it as you want. Using these methods allows you to put columns in the Structural Column category so they still take on some of the behavior of a totally vertical column.

In-Place Family Creation

In-place families can be used for columns that are only specific to the project. This method will give you the ability to define the sloped column by using the Solid and Void Extrusion tools in the In-Place Family Editor while viewing the modeled components in your project around it. The same tools used in an external family can be used when implementing this method. The basic procedure for creating an in-place structural column family is as follows:

1. While in your project, select the Create tool from the Modeling tab.
2. Select the Structural Column category.
3. Assign it a name specific to its purpose.
4. Use the Solid and Void tools to create the shape of the column.
5. Add any model or symbolic lines that you need.
6. Add new subcategories if needed, by choosing Settings ➢ Object Styles. Click New and create a new category under Structural Columns. Make sure that the Subcategory Of option is set to Structural Columns.
7. After you've created a subcategory, you can assign any solid shape to it by opening the shape's properties and selecting your new subcategory from the Subcategory field in the Identity Data group. Voids cannot be assigned a subcategory.
8. Assign detail level visibility by selecting solid or void shapes and clicking the Visibility button on the Options bar.
9. Close the In-Place Family Editor.

If you plan to create multiple columns with this method, you might want to create all of the columns inside one family. Each instance of an in-place family that you copy will create another in-place family. If you have to change something, you will have to change them all individually or delete all but one and recopy them back into place. Be careful when deleting columns because everything associated with them will be deleted too. Creating them all in the same family can also eliminate extra copies that tend to clutter up your Project Browser.

A Group Is Better than One

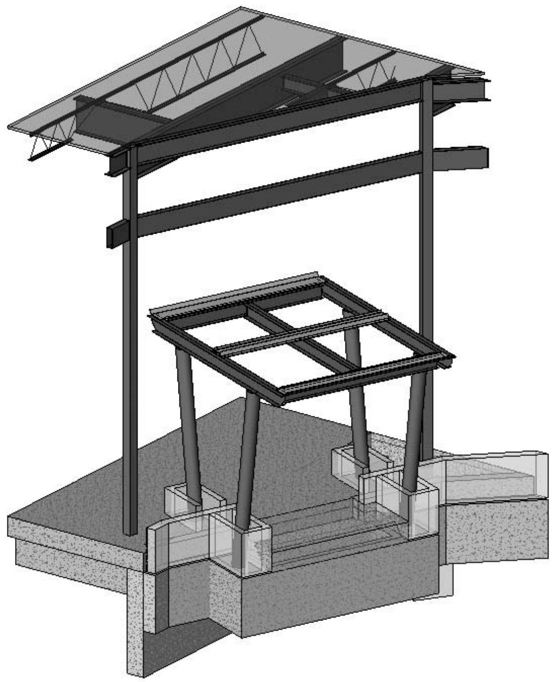
Before copying the same column over and over inside the family, create a group. Select the column element(s) and choose Edit ➢ Group ➢ Create Group and give the group a name. Once you've created and copied the group, you can make changes to one of the columns and all of the others will change with it. See Chapter 18 for more information on grouping.

Using in-place families is a quick way to get the geometry you need inside the model. These families work well when you have unique situations. When things tend to change, it is a good idea to try to build a bit more flexibility into the family so that it reacts to those changes quickly. Using external families is more flexible, and they can be used in other projects.

External Family Creation

You can use the same methods to create an external family, except these families are created outside the project. External families allow you to include a bit more flexibility. They still have the same issues in the way that Revit Structure interprets them as physical and analytical objects. Figure 4.19 shows an example of a sloped column created as an external family and loaded into the project to be used in a vestibule entrance.

FIGURE 4.19
External sloped column family used in a project



If external families are created properly, they can be reused in other projects to speed up the modeling process. When building this flexibility into the family, keep to the basics. Focus on building flexibility into the family for when columns change size or location and taking into account the various slopes that may occur. Some things to think about before creating this type of family are:

- How should the angle of the slope be determined?
- Will you know the angle of the slope, or will you know the horizontal offset from the center of the bottom to the center of the top of the column?

- What type of material will you use?
- Will you use a blended extrusion or a sweep? If it is a sweep, should you use a sketch or load a profile that will let you easily change sizes?

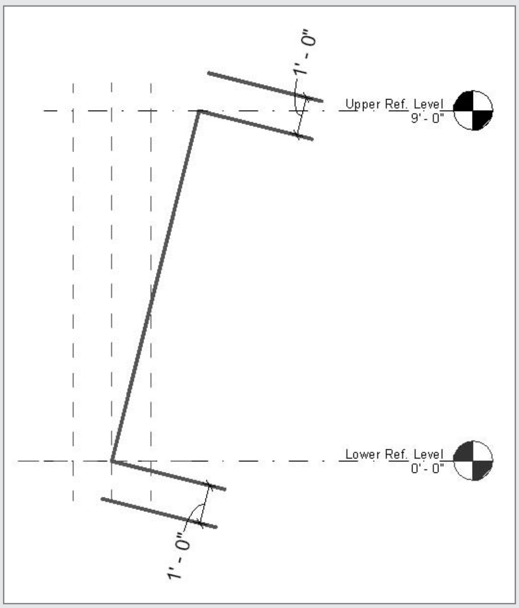
All of the methods we've discussed will allow you to create sloped columns in your project. Knowing the pros and cons of each method, understanding the behavior each one provides, and anticipating how they need to be used in all views throughout the project will help you choose the right process.

Exercise: Creating an External Sloped Column Family

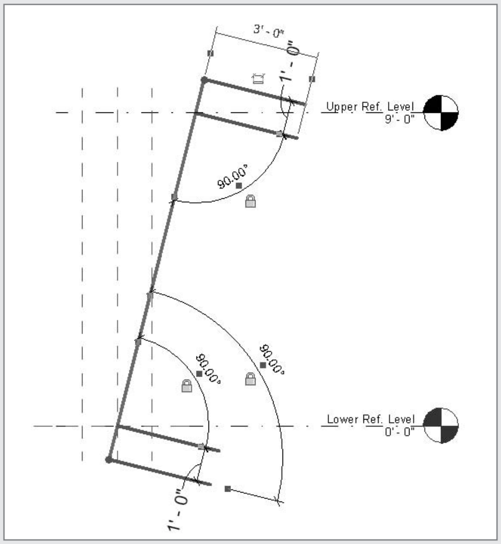
In this exercise, we will walk through the basic steps to creating a sloped column. We'll use a Structural Column template, and the slope column will get its slope from a defined angle.

Build the Skeleton of the Family

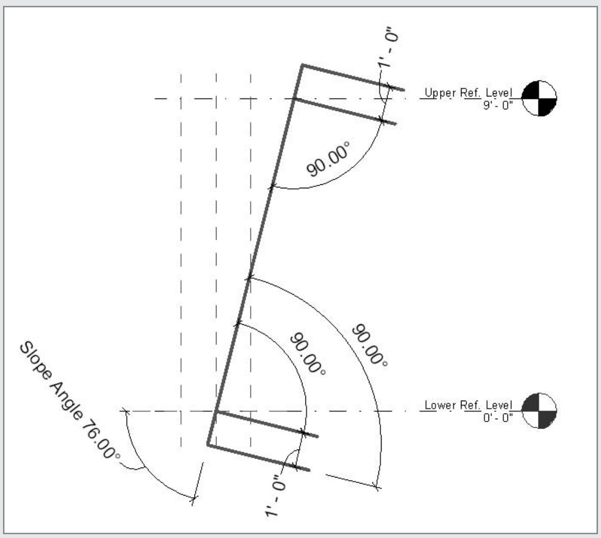
1. Choose File ➢ New ➢ Family.
2. Browse to `Imperial Templates\Structural Column.rft`. Select the file to open it.
3. Choose Settings ➢ Families Category and Parameters.
4. Set the Structural Material Type option to Steel.
5. Set the Beam Cutback in Plan option to From Geometry.
6. Activate the Elevation Front view, and create five reference lines, as shown in the following illustration. Place the first one at a diagonal from where the center reference plane intersects Level 1 and up to the right, so you select Level 2. Place two more reference lines perpendicular to the one you just placed at the top and bottom.



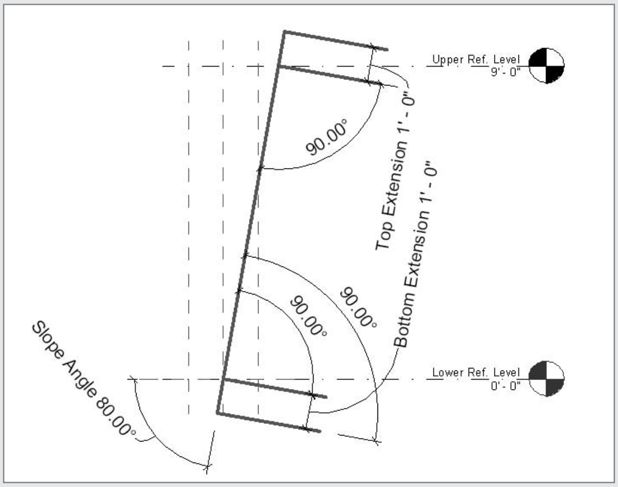
7. Click the first diagonal reference line that you placed and drag the ends of it by selecting the blue dots so that it extends to the farthest placed perpendicular reference lines, as shown here. Place angular dimensions at 90 degrees and lock the dimension.



8. Place another angular dimension (as shown in the following illustration) defining the slope of the column. After the dimension is placed, select the dimension and select Add Parameter from the Label pull-down in the Options bar. In the Parameters Properties dialog, give the parameter a name of **Slope Angle**, make it an instance parameter, and group it under Dimensions. Click OK to close the dialog. You've just created a parameter that allows you to change the slope angle.



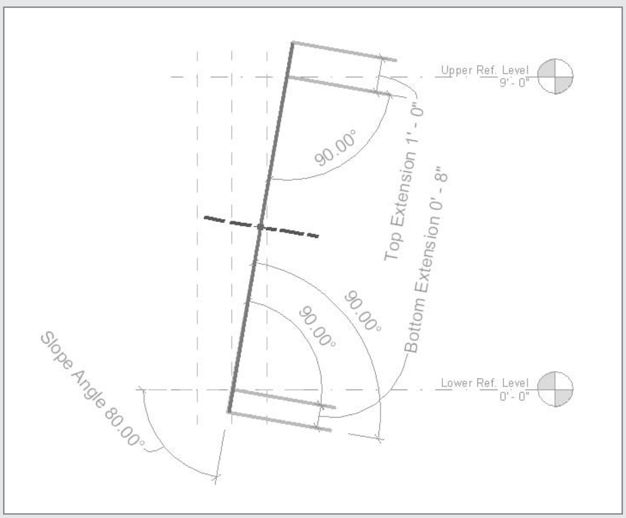
9. Create an instance parameter for the two 1´-0˝ dimensions, as shown here. Call one **Bottom Extension** and the other **Top Extension**. Group both of them under Dimensions. These parameters will allow adjustments to be made to the extents of the top and bottom of the column.



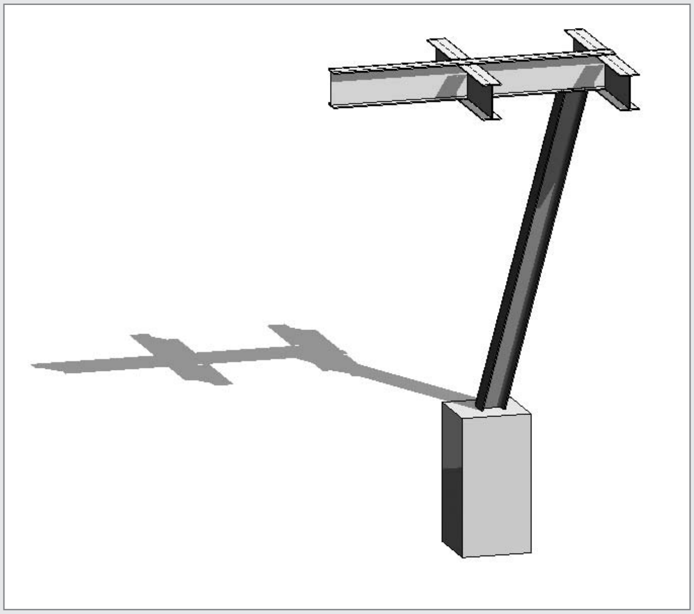
10. Select Family Type from the Family tab on the Design bar and flex the reference lines and parameters that you added to verify that everything is working as expected. The slope angle should rotate the reference line while maintaining the top and bottom extensions. Once everything is flexing properly, you can move on to adding geometry to the family.

Add Geometry to the Family

1. On the Family tab, select Solid Form, then Solid Sweep.
2. On the Sketch tab, click Pick Path. Select the diagonal reference line shown in the following illustration. Select Finish Path from the Pick Path tab. By using Pick Path, you are automatically locking the path to the reference line. If the reference line moves, the path will move with it.



3. On the Sketch tab, select Profile and then select Load Profiles from the Options bar. Browse to the `Imperial Library\Profiles\Structural` folder and select the profile for the shape of the sloped column. In this case we will be using the `W-Wide Flange-Profile.rfa` from the Steel folder and selecting a W8X31.
4. On the Options bar select the By Sketch pull-down and change it to the Wide Flange-Profile:W8X31 profile shape. Also on the Options bar, change the rotation to 90 degrees. Click Finish Sweep on the Sketch tab.
5. Flex the family once again to verify that the geometry is moving with the reference lines/planes or what is also known as the skeleton of the family.
6. Save the family if you want and load into a project.



Since we used a solid sweep and a profile shape, you can add profiles of different shapes to the family. You can add a Family Type parameter set to grab from the Profiles category to switch the different profile shapes while the family is loaded into your project.

The top and bottom extension parameters were added to give a bit more flexibility for the top and bottom connections. If needed, the ends of the column can be extended far enough to allow you to cut the geometry to a reference plane to create a horizontal cut on the bottom or top of the column. You can also create void forms inside the family to cut the end shape of the column.

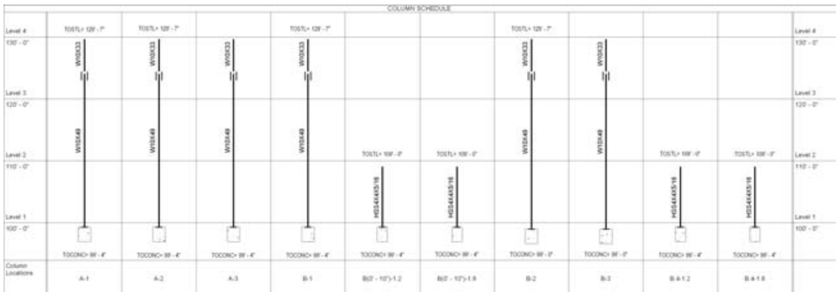
To help with the display of the family when loaded into a project, you can set visibility options as well as add symbolic line work. You can create a Material parameter and link it to the geometry, and then create a subcategory and link it to the geometry so it can be displayed on or off as required.

As you can see, you can create sloped columns rather quickly and add them to your project. Giving your column its finishing touches can be just as easy once you know how to work inside a family and create parameters. The amount of flexibly needed will depend greatly on how the family will be used. Don't build a lot of flexibility into the family if it does not need it. Chapter 18 will discuss family creation in greater depth and explain how you can add functionality to your families.

Using a Graphical Column Schedule

A Graphical Column Schedule (GCS) is extremely useful for calling the size, reinforcing, and connection information out for columns on a multistory structure in an elevated graphical display rather than the standard method of using a mark number and text-only schedule. Revit Structure automatically keeps track of any structural column that is placed in your project and links it to a grid intersection. If it is not on a grid, Revit Structure will link the column to the nearest grid. Not only can the GCS shown in Figure 4.20 be used to document the information pertaining to your structural columns, but it can also be used to keep track of information while you are working and help maintain the integrity of your model.

FIGURE 4.20
A Graphical Column Schedule showing the top and bottom of steel elevations



Since GCSs are a completely different schedule than Revit Structure's Standard Schedule, they get their own category in the Project Browser called Graphical Column Schedule. (We'll discuss how to create a Standard Schedule in Chapter 11.)

The basic procedure to create a GCS goes like this:

1. Select View ➤ New ➤ Graphical Column Schedule, or in the View tab on the Design bar click Graphical Column Schedule.

Note that if gridlines are not present in your project or the 3D extents of at least two grid intersections do not cross the plane of a structural column, you'll see the warning "No columns are joined to grid lines, or view parameters exclude all columns." Revit Structure will create a new GCS view but will not generate a schedule displaying column information.

2. Right-click the new GCS view in the Project Browser and select Properties, or right-click anywhere in the view itself and select View Properties.

3. Make changes to the view's properties to give it the final look you want.
4. Once the GCS is placed on a sheet, change settings to allow you to split the schedule onto multiple sheets. (You can learn more about placing GCSs on sheets in Chapter 12.)

With the GCS, you have access to only a few settings that will allow you to tweak the appearance of the schedule and control the type of columns that will show up in the schedule. There are other approaches that allow you to control the display of the graphics in the schedule, as you'll see in the next section.

Setting the Appearance

In the Element Properties dialog of the GCS (see Figure 4.21), you will find most of the settings for adjusting its appearance and how the information in the schedule displays. We will step through a few of the primary controls to show you how they affect the schedule's display.

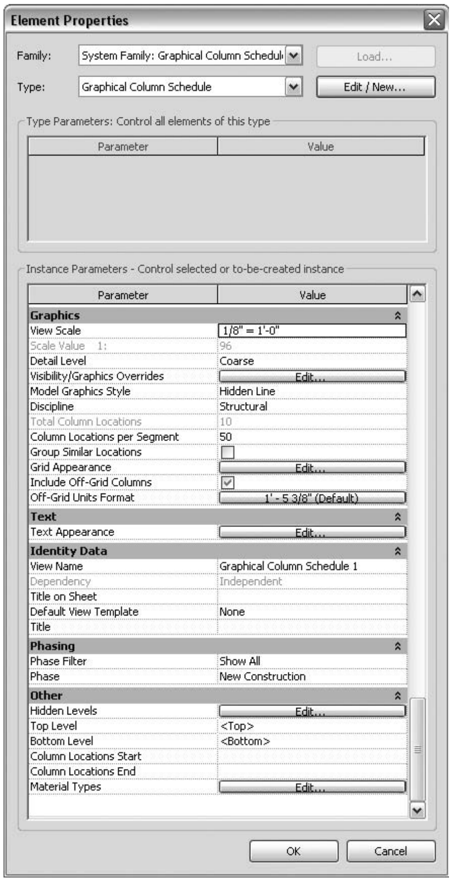


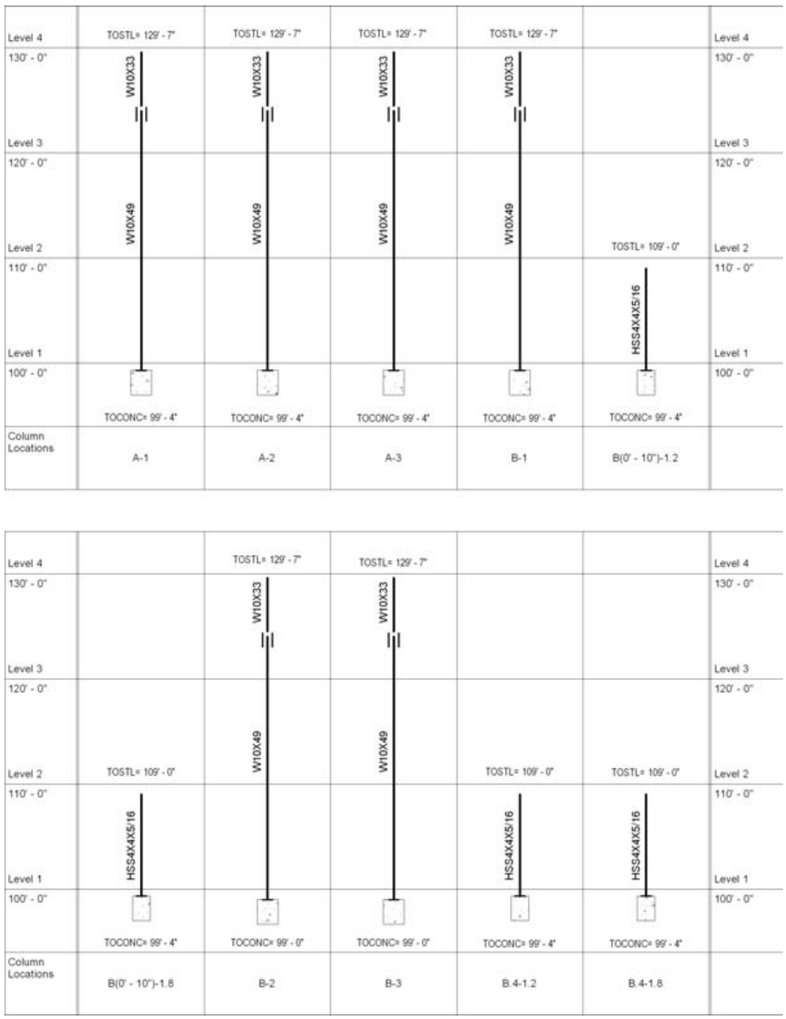
FIGURE 4.21
Changing the properties of a Graphical Column Schedule

The GCS has the same basic properties of any other view that Revit Structure creates, and the properties will behave pretty much the same. You have View Scale, Visibility/Graphic Overrides, Detail Level, and Discipline options. If you set Detail Level to Coarse, Revit Structure will display steel shapes as symbolic lines and include symbol representation for the top and base of column connections. You can apply view templates to the schedule, and it can have its own title separate from the title on the sheet.

Column Locations per Segment

Some projects have several grid intersections with columns, which can make your schedules quite lengthy. For this reason, you can set the number of vertical rows to display in the schedule (Figure 4.22) before the schedule automatically splits into a new segment (it remains part of the same schedule). When you modify Column Locations per Segment, specify the maximum number of vertical rows you want to display.

Figure 4.22
A GCS split into five segments with a vertical height between them of ½″



Group Similar Locations

Select the option Group Similar Locations to combine columns that have the same information into the same vertical row (Figure 4.23). The Group Similar Locations option can also be used to change the properties of a group of structural columns all at once. This approach may be easier than selecting columns one by one and changing their properties.

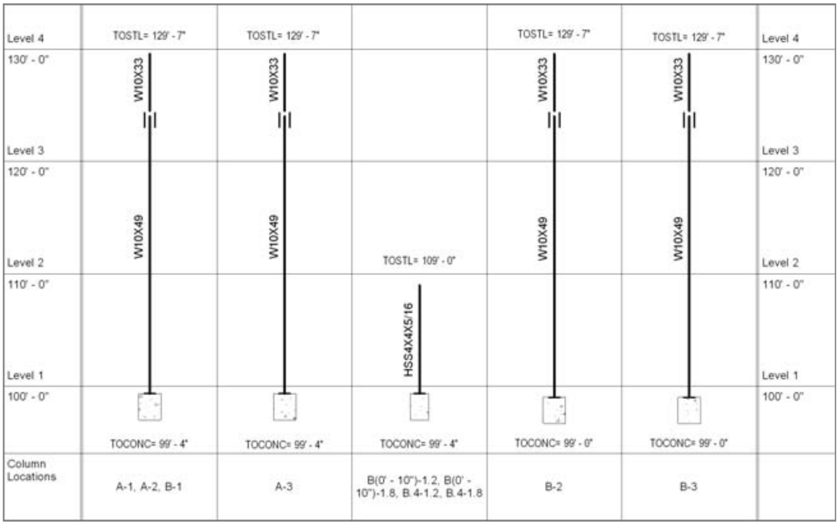


FIGURE 4.23 A GCS with Group Similar Locations checked can eliminate redundant cell values.

Grid Appearance

The Grid Appearance tab, shown in Figure 4.24, appears when you click the Edit for the Grid Appearance parameter in the Element Properties dialog for the GCS. You may have to adjust the Horizontal Widths options of the vertical rows to accommodate level names that are too long to display legibly in the schedule. Perhaps you have to widen the vertical rows where the graphics of the structural columns display to provide adequate room for text in tags.

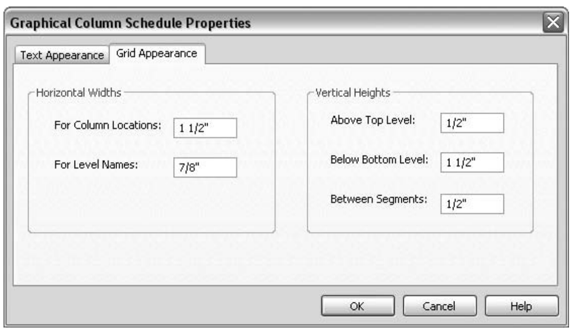
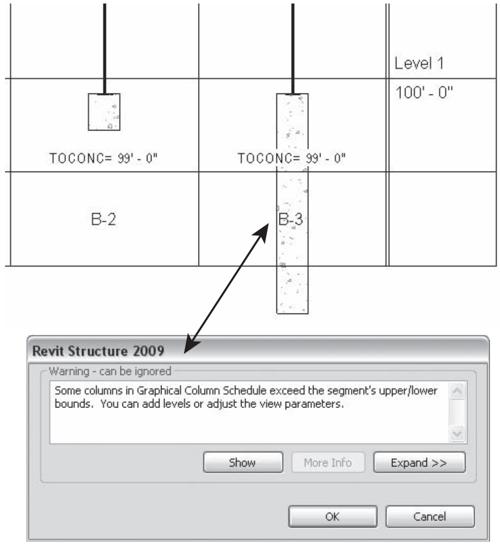


FIGURE 4.24 Setting the grid appearance of the GCS

You may have to adjust two of the Vertical Heights options, Above Top Level and Below Bottom Level, when structural columns exceed the GCS's upper or lower boundaries. As Figure 4.25 shows, the columns in the GCS are shown at their modeled size and length when a column is dropped well below Level 1 for a utility pipe.

FIGURE 4.25
When columns exceed the segment's lower bounds of the GCS, adjust the Vertical Height of the Below Bottom Level option.



The last setting in the Vertical Height options is called Between Segments. Use it to set the dimensions between columns that are broken up into multiple segments, as shown earlier in Figure 4.22. GCSs are similar to other views in that they cannot be put onto the same sheet twice. Breaking them into multiple segments will shorten the overall width, thus allowing you to put more than one schedule on a sheet while keeping all information inside one schedule.

THIN LINE = GCS OUTLINE AND GRIDLINES

You can set the line weight thickness of the Graphical Column Schedule's outline and gridlines by opening the Visibility/Graphic Overrides dialog for the view and selecting the Model Categories tab. Expand the Lines category, and for the subcategory Thin Lines, modify the Project Line value.

TEXT APPEARANCE

The Text Appearance tab, shown in Figure 4.26, appears when you click the Edit button for the Text Appearance parameter in the Element Properties dialog for the GCS. You have seen several images in previous sections that show ways you can display the line work and column information for a GCS. Here you will find that you can also make changes to the automatic text that Revit Structure displays in the schedule.

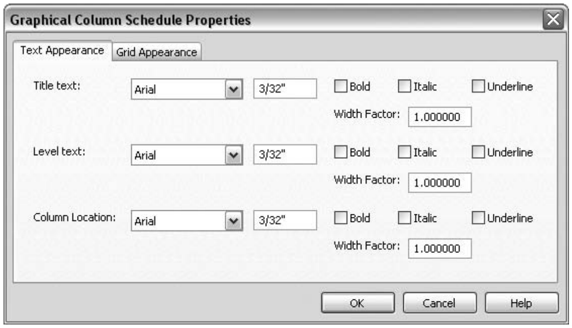


FIGURE 4.26
Setting the text appearance of the GCS

Another way to fit level names that are too long to display legibly in the vertical level row of the GCS is to adjust the text's Width Factor value. Also when you are grouping similar columns, you can do this in the Column Location field. This allows you to fit more grid intersections in the box without having to adjust the width of the grid intersection location row. You can set other text formats, such as bold, italic, and underline, to help achieve the look you want.

HIDDEN LEVELS

Clicking the Hidden Levels button in the Element Properties dialog of a GCS takes you to the Levels Hidden in Graphical Column Schedules dialog shown in Figure 4.27. Here you can turn off the display of a level by clicking its check box. All levels will appear in the schedule regardless of their 3D extents. For example, if you have created levels in your project for the sole purpose of tying together geometry or have levels for both top of slab and steel, select these levels so that Revit Structure will hide them in the GCS.

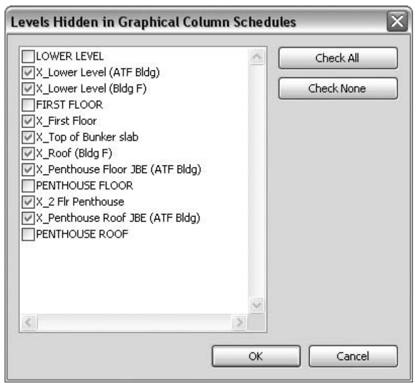


FIGURE 4.27
Selecting levels you want to hide in the Graphical Column Schedule

TOP AND BOTTOM LEVEL

By default, Revit Structure displays the lowest (bottom) and highest (top) levels in the GCS. You can modify the Bottom Level and Top Level settings to display only columns between a specific set of levels.

For example, suppose you have all the columns modeled on your project and you have to issue a Foundation package. You want to include structural column information for only the

first lift of columns to obtain dowel reinforcing or anchor rod information. Set Bottom Level to Level 1 and Top Level to Level 2. With these settings, the GCS will only show column information for columns between Levels 1 and 2. When the next issue comes and you need to include the additional column information, you can change the Top Level setting back to the default.

Here's another example: suppose you want to schedule only the Penthouse Columns on Level 10. To do so, set Bottom Level to Level 10 and Top Level to Penthouse Roof.

Column Location Start and End

The Column Location Start and End option works in a similar way to the Top Level and Bottom Level settings, except that it uses the grid intersections. You can specify a grid intersection to start from and one to end with.

For example, suppose you have three zones, Zone A, Zone B, and Zone C, and you want to separate your GCS into three different zone-specific schedules. This separation helps anyone reading the documents to easily find the grid intersection. For each GCS, set the Column Location Start and End option to only show the extent of grid intersections for each particular zone.

Once you have the overall look of the GCS in place and you have it set up the way you want, you can move on to adding annotations to display structural column properties that you need for documentation, or to put information into the model.

Annotating

The biggest display of annotations in the GCS is the tagging that pulls information from the column's properties. Any information that is part of the column's properties can be displayed with a tag that has a direct link to a particular parameter. Another form of annotation is the use of spot elevations for indicating the top or bottom of column elevations. A third form is, when the View option is set to Coarse, steel shapes display symbolic connection symbols.

These three annotation methods automatically adjust when the GCS changes form or when columns shift within it. Tags and spot dimensions maintain their relationship to the columns as other columns are added to the schedule, forcing them to shift down the line. You can add normal text as annotations, but this text will not move as the schedule changes. You will have to visually check the text for proper placement and manually move it.

Tagging

Most of the parameters that you tag are either project or shared parameters. When creating such parameters, you must specify Instance or Type. You establish each parameter depending on the type of project and the various stages of packages you need to issue.

Figure 4.28 shows a typical steel column, which can display information such as:

- Size of the member
- Base plate size
- Anchor rod size
- Embedment length
- Layout type

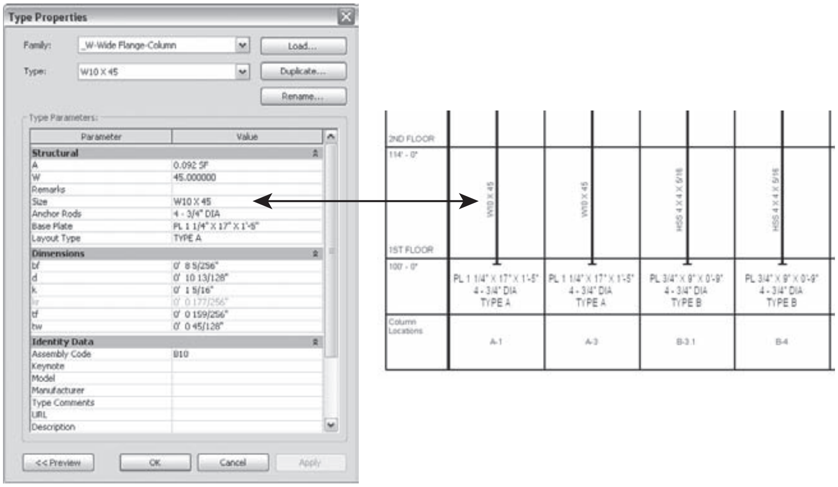


FIGURE 4.28 Placing text parameters into steel column families allows you to create tags to display base plate information in a GCS.

Figure 4.29 shows a typical concrete column, which can display information such as:

- Size of the member
- Vertical reinforcing
- Tie reinforcing
- Layout type
- Dowel reinforcing at foundation

You add text parameters for this information to the structural column families. Tags pull these values to display in the GCS.

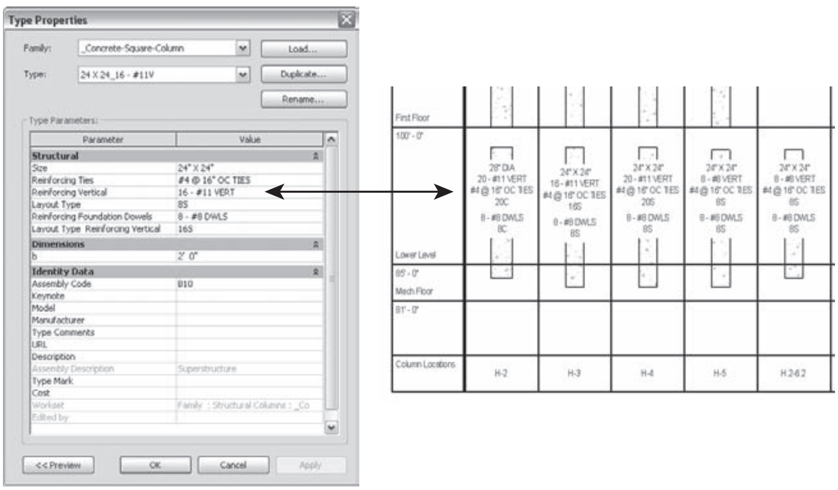


FIGURE 4.29 Placing text parameters into concrete column families allows you to create tags to display reinforcing information in a GCS.

Wide Flange Columns Don't Have Tie Reinforcing

When adding text parameters to columns as project parameters, you can only select the Structural Column category. This means that you may be adding concrete reinforcing parameters for concrete columns, but they are also going to show up in any steel, precast, or wood columns that you have in your project. Creating shared parameters and placing this specific information directly into the family itself prevents unnecessary information from showing up in the column's properties.

Spot Elevations

You can add spot elevations to the GCS by selecting the Drafting tab on the Design bar or by choosing Spot Dimension ➢ Spot Elevations. As Figure 4.30 shows, not only can you use spot elevations to indicate the top and bottom elevations of columns and any other elevations that you need for documentation, but you can also use them to help control the placement of the column geometry in the model. A GCS you create simply to help control the placement of the column geometry and serve as a model consistency check can be maintained easily.

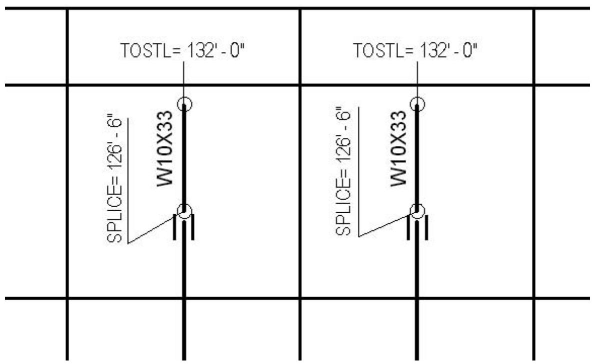


Figure 4.30
Using spot elevations in a GCS can help you annotate live information from the modeled geometry.

Splice and Base Plate Symbols

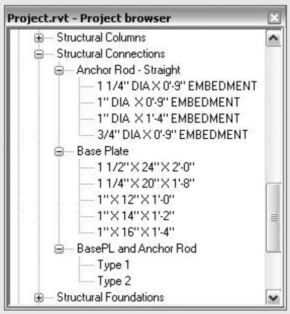
You know that Revit Structure displays symbolic symbols for steel shapes and their connection types in other views when they are set to a coarse detail level; the GCS does the same thing. Chapter 2 describes in great depth how Revit Structure controls these symbols for a structural column and how to create new ones as well as set their display for a project. Since structural steel is usually shown with single symbolic line work, these symbols are only available when you're working with steel shapes or families that have Structural Material set to Steel. If available for a structural column, these symbols can be found in the Graphics group of the column's properties. Making use of their display in the GCS will put the finishing touches on a well-displayed and well-documented schedule.

Modeling Base Plates

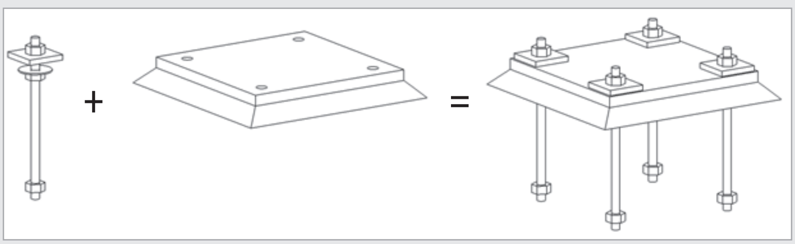
Revit Structure offers families for placing base plates as modeled geometry into your project. You can load these families from the `Imperial Library\Structural\Connections` folder. These families are face based and are part of the Structural Connection category. We attempt to take these families to the next level so we can schedule them and so they can serve as an interference check for the architect when plates and bolts tend to project outside a wall or column enclosure.

We continue to experiment with placing base plates as modeled geometry in each project, along with using text-based parameters in schedules for our documentation. Typically each new version lets us take this concept a bit further. Modeling base plates is probably not a wise approach for all your projects, so do it selectively. Keep in mind that modeling this amount of detail in a large project can decrease the performance of the model. We feel that eventually the software will catch up to this level of detail, so we are trying to dig in at whatever level we can to keep the BIM method of thinking going.

For example, we decided to develop an Anchor Rod and a Base Plate family, and set each as shared in the properties. We nested the new families inside the Base Plate Connection family so that we can place them into a project as base plate connections. Since we specified these families as shared, they show up as individual families under Structural Connections in the Project Browser, as shown here. Therefore, we can easily duplicate them and create new sizes as needed.



In the Base Plate Connection family, we can select the anchor rod and base plate types (using a Family Type parameter) to create base plate connection types, which we can then schedule. These families eventually can become attached to columns when we place them into a project. Here is the overall look of the concept.



This is now a pretty advanced family and reacts well when properties of the elements change. When the anchor rod size changes, the grout changes with it, and the holes in the base plate change in size accordingly. Also, when the base plate changes size, the anchor rods and its components change with it. Visibility and subcategories are also assigned to the different components. This way, only the base plate displays at the medium detail level; everything shows up at the fine detail level. We can turn off components in the Visibility/Graphic Overrides dialog of the view from the subcategory setup.

This is a concept that we continue to look at, and the process still has a few kinks we need to work out. A few questions come up. For example, should this base plate, grout and anchor rod be part of the column family, or should it be a stand-alone connection as it currently is? What happens when there are more than four anchor rods? We hope that the ability to push the limits of Revit Structure in this way will eventually become part of the GCS.

Displaying Structural Columns

As you can guess by the name Graphical Column Schedule, all columns displayed are graphically shown as their geometry. Earlier we discussed setting the appearance of the columns: we explained how you can set a steel column to display as a symbolic line, and you learned that concrete columns are usually shown as double lines. Now let's discuss what you do after the columns are actually in the schedule.

What determines that a column is put into the schedule? What do you do if columns don't show up in the schedule? What happens when you don't want certain columns to show up in the schedule? These are questions that you are going to want answered when you're working with a GCS. Knowing these answers will help you create your schedules quickly and ensure that only the desired columns display.

Structural Columns That Are Offset from the Grid

Revit Structure gives you the ability to display only those columns that are on grids or those that are offset, and it still gives those columns a location mark based on the grid names that are intersecting them. Revit Structure creates the location marks automatically using a set of rules. In some cases, the program tries to be flexible in allowing you to choose what is displayed. After determining the best grid to reference, Revit Structure displays this value in the properties of the columns, under the Constraints group in the Column Location Mark parameter. If more than one grid can be referenced to the offset column, Revit Structure allows you to select which column location mark you want to display. Figures 4.31 and 4.32 show that the read-only behavior of this parameter changes depending on where the column is located.

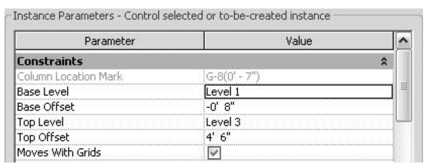


Figure 4.31
The column location mark sometimes displays as a read-only value.

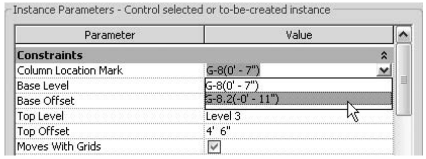


FIGURE 4.32
The column location mark removes its read-only behavior to allow you to select a different location mark.

Automatically scheduling offset columns, determining their location mark, and giving the user a bit of flexibility can be complex. You may have to resort to other methods to get the GCS to perform the way you want it to.

Automatic

In the properties of a GCS, you can choose Include Off-Grid Columns. With this option checked, all columns—regardless of their placement on a grid—should be placed in the GCS as long as the 3D extents of the grids are crossing the horizontal and vertical plane of the column(s). Revit Structure will continue to give the offset column a location mark referencing the closest grid intersection based on a set of calculations.

Figure 4.33 shows that if an offset column's bounding box (the extent of its geometry) intersects or its edges touch a grid, its location mark will reference the closest grid intersection.

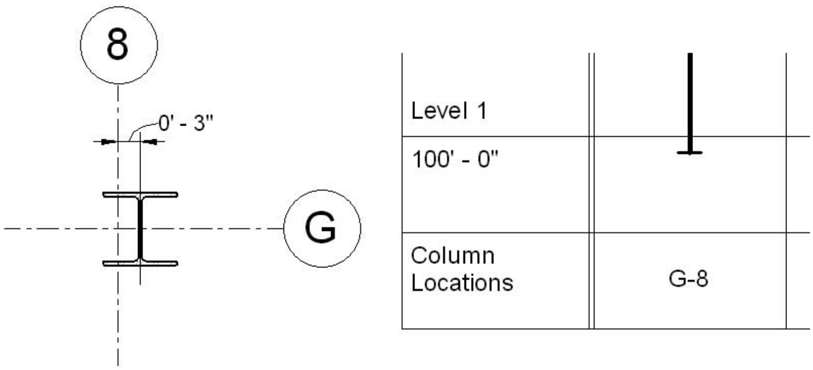


FIGURE 4.33
Offset columns with a bounding box intersecting the grid have a normal location mark.

Figure 4.34 shows that if an offset column's bounding box (the extent of its geometry) does not intersect or its edges do not touch a grid, its location mark will reference the closest grid intersection, with an added annotation indicating what the offset is from the referenced grid.

Also included in the properties of the GCS is a setting called Off-Grid Units Format. This setting, which behaves just like any other unit format overrides (similar to dimensions), is located just below the Include Off-Grid Columns parameter. This override allows you to have some control over the display of the offset dimension value that is placed after the grid mark in the Column Locations row.

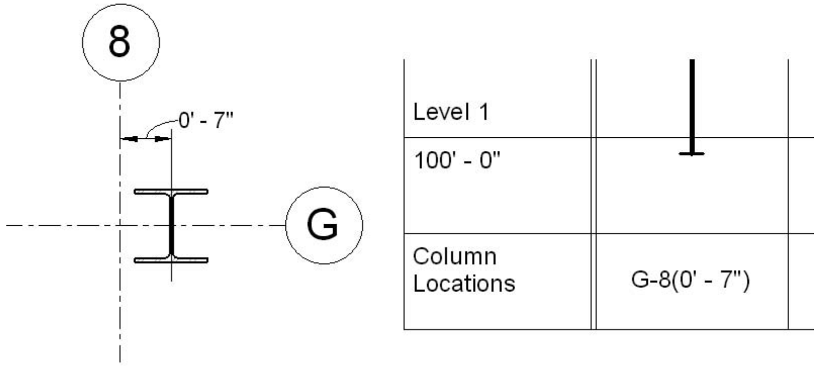


Figure 4.34
Offset columns with a bounding box not intersecting the grid are noted with an added dimension in the location mark.

> **What Determines the Offset Column Location Mark?**
>
> Revit Structure calculates which grids the offset column is closest to and assigns the column those grids for a location mark. To learn about these calculations, in Revit Structure's Help guides search for "Determining off-grid column location marks."

How the GCS displays and annotates offset columns may not be to everyone's liking. Sometimes a quick duplication and modification to an already created structural column family will do the trick. If that doesn't quite work, you might want to try creating extra grids that will keep the column on a grid intersection. Any of these methods will work, but you will have to decide which one is best for each particular project.

Modifying the Family

In the next method we'll discuss how you make a few quick edits to an already existing column family (or if you are creating your own family, keep these edits in mind). In short, you will make changes to the column family that allow Revit Structure to shift what it considers to be the center of the column. Make sure that you are not making changes to the normal Structural Column family, but that you save this family under a different name using a suffix of "-Offset" or something similar. One of the rules for displaying a structural column in a GCS is that it must be centered on a grid intersection (unless you have the Include Off-Grid Columns parameter checked). We'll make a few quick edits to a duplicated Structural Column family so it has instance parameters to adjust the location of the Center (Left/Right) and Center (Front/Back) reference plane inside the Structural Column family. The basic procedure looks like this:

1. Edit or open a column family that will be uniquely named for its offset behavior.
2. While in the Lower Ref. Level, create a new reference plane parallel to both of the Center (Left/Right) and Center (Front/Back) reference planes.

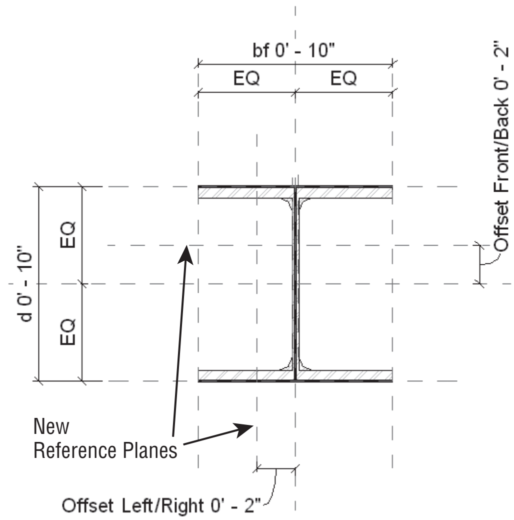
You can find these reference planes by viewing their properties and looking at the Is Reference value. These reference planes should currently be placed at the center of the column.

3. In the properties of the new reference planes, reassign the Is Reference values to the appropriate center name in the pull-down menu and select the Defines Origin check box.

This step will automatically set the old reference planes located at the center of the column to Not a Reference and set the origin of the column to the new reference planes intersections.

4. Place a dimension between both of the reference planes, as shown in Figure 4.35, and add an instant label to them defining the column offset.

Figure 4.35
Redefining the column centerline reference planes and origin in a Structural Column family



After loading the family into the project, you can use your new family when you want to display an offset column. If using this method is not feasible, consider using another grid type as a centerline and using a unique grid name mark to display in the GCS.

Why the Workaround?

Revit Structure automatically schedules offset columns in the GCS. You also have the option to not display the offset columns in the schedule. But like anything else, there will be cases where Revit Structure's automatic methods do not perform to your standards. In such cases, you can resort to other workarounds that go beyond the current capabilities of Revit Structure.

Using a Grid as a Centerline

In our next method, you create a new grid type by duplicating your standard one and naming it something like **Centerline**. Set its Symbol parameter value to None so you will only see the gridline when you place it in your project. Assign a color to it so it visually stands out from your normal grid. Place your offset columns on this gridline. Since it is still a grid, you will have to give it a name. If the grid you want to associate it to is named Grids C-2 and its offset is from Grid 2, then you could name the Centerline **Grid C*** or something similar. It would then show up in the GCS as C*-2, as shown in Figure 4.36.

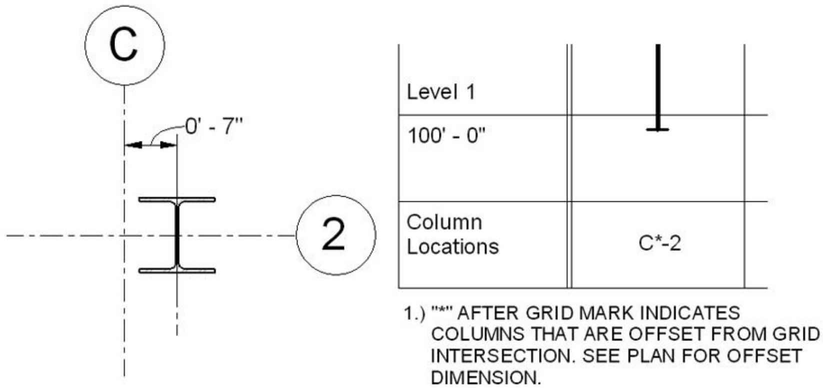


Figure 4.36
Creating a new grid type along with a unique grid name to schedule offset columns

You can add a note to the bottom of the schedule that states something like this: "Columns shown in schedule with '*' after the grid name indicates columns that have been offset from a grid." This approach allows you to see the offset centerline in all views because it behaves as a grid; since the grid type is not assigned a symbol, the bubbles will not display either. An offset column usually needs a centerline to denote its offset dimension, which makes this method easy because you don't have to place a separate detail line in every view in which you need to show the column.

Why Don't Columns Show Up in a GCS?

There are times when a column appears to be on a grid intersection but doesn't appear in the GCS. Even if a column isn't on a grid, it should at least show up if you've selected Include Off-Grid Columns in the properties. If this is the case, you might want to check the 3D extents of the grids. In order for a column to be referenced to a grid intersection, at least two intersecting grids' 3D extents must cross the column's plane. The 3D extents might cross in the horizontal (plan) direction, but might not be pulled high or low enough to intersect its plane in the vertical direction. It could also be the other way around. Remember, grids have a 2D and a 3D extent. The 2D extent is specific to the view in which it is being displayed. Cut a couple of quick sections to see if this is the problem.

Using Columns Structural Material Type

Earlier we discussed setting the Structural Material property for a Structural Columns family. Not only does this setting dictate how members frame to the column or how it displays in views, but it can be used to display or not display columns in the GCS. To use the Structural Material property to display columns in a GCS, first open the GCS's Element Properties dialog box. Under the group Other, you will find an Edit button. Click this button to open the dialog box shown in Figure 4.37. By default, Revit Structure has all five material types checked. Deselect any you don't want to include in that GCS.

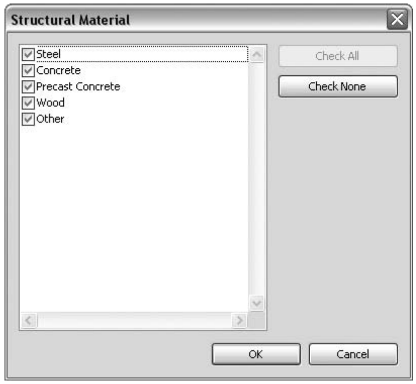


Figure 4.37
You can remove columns from the GCS based on their structural material type.

This method of displaying columns can be useful when you have certain columns with material types that you don't want to appear in the GCS. Maybe you have a few wood columns scattered throughout a project, or a concrete structure that has a few steel columns that will be scheduled in a Basic Schedule. Deselecting those structural material types will remove them from the schedule.

If you want only a portion of those steel, wood, or concrete columns removed, you will have to apply phasing or some additional filters to remove them in a more selective way.

Applying Phases

Applying phases to a GCS works the same way that it does when applying them to other views. The GCS provides a phase and a phase filter that you can apply to help drive which columns will be displayed. We'll discuss phases in much greater depth in Chapter 16. The same methods and procedures discussed in that chapter can be applied to the GCS.

You can use phases to help control the visibility of existing columns in the GCS. This is probably the easiest way to remove them from the schedule (as long as are assigning these elements to the Existing phase and you set the Phase Created parameter to Existing). Follow these steps:

1. In the properties of the existing column, make sure the Phase Created parameter is set to Existing.
2. Right-click in the GCS's view and select View Properties, or right-click on the GCS's name in the Project Browser and select Properties.

3. Set the value of the Phase Filter option located in the Phase group to Show New. Click OK to go back to your GCS.

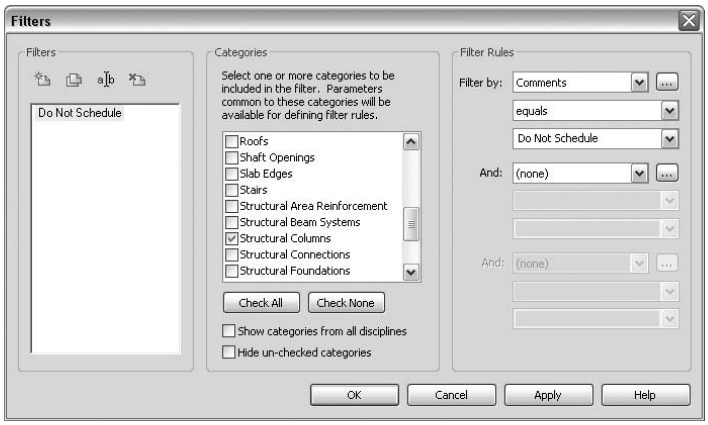
4. Those columns for which you set the Phase Created parameter to Existing have disappeared from the schedule.

You can use phases to display columns in the GCS depending on the project stage. You may even create schedules just to see how columns are modeled or to reveal the various stages at which they will be issued. When using phasing does not quite get you 100% satisfaction, you can apply filters, as you'll see next.

Applying Filters

Filters let you be a bit more selective in determining what columns show up in the GCS and—in some cases—*how* they display inside it. Filters can be a selection of elements that you save or a set of rules that you put in place to determine what is put into a selection set as your model progresses. Figure 4.38 shows a Generate by Criteria filter called "Do Not Schedule" that does the following: it finds all the project elements from the Structural Columns category and if the value of an element's Comments parameter equals Do Not Schedule, then that element becomes part of the selection set. As this parameter value changes, columns continue to be added or removed from the selection set. This can be a much better method than just saving a selection set and manually having to add and remove elements as they are modeled in your project.

FIGURE 4.38
Using a Defined by Criteria filter allows you to selectively display columns in a GCS.



After your filter is created, you will have to apply it to the GCS view using the Visibility/Graphic Overrides dialog box.

To make a Selection Set filter, perform the following steps:

1. Select the elements that you want to be part of the set.

2. Choose Edit ➢ Selections ➢ Save Selection.

3. Give the Selection Set filter a name that reflects its purpose.

Next you apply this filter to a view in the Visibility/Graphic Overrides dialog box.

To create a Defined by Criteria filter, follow these steps:

1. Choose Edit ➢ Selections ➢ Edit Selections.
2. In the resulting dialog box, click the New button to create a new filter.
3. Select the Define Criteria option and assign your filter an appropriate name.

Specifying the selection criteria is similar to the Basic Selection Set filter in that you can manually select any elements regardless of their categories or parameter settings.

4. Apply the appropriate settings (such as the categories that the filter will apply to), and then specify any rules that will apply to the filter.

After you save your filter, you can add or remove elements or change the filter rules (choose Edit ➢ Selections ➢ Edit Selections). Select the filter and click the Edit tab. If the filter is a Define by Criteria filter, it will display the Filters dialog box, where you can change its rules. If it's a Selection filter, you will be put into a Selection Edit mode, and the Design bar will show options for adding or removing elements from the selection.

To apply a filter to a GCS view, perform these steps:

1. While in the properties of the GCS view, click the Edit button next to the Visibility/Graphic Override parameter.
2. Select the Filter tab at the top of the dialog box to access the filter settings.
3. Click the Add button to add an already created filter or click Edit/New to create a new one.
4. After you add the filter on the Filter tab, you can make changes to the filter's elements.
5. Deselecting the Visibility option will remove any structural columns that are part of the filter from the GCS. You can also apply other override settings to change the look of the graphics of the scheduled columns.

Filters can be powerful tools for developing GCSs that will help you document your model. In some cases, you can use existing parameters such as the Comments field, or you may need to create your own. Filters can be used to help create separate GCSs for zones of a building.

Adding a parameter called Scheduled Zone will allow you to give each structural column a property indicating the zone the column belongs to. You can use a filter to select only those columns in a particular zone. Each time you create a GCS, those filters are applied to show only the columns for a particular zone.

Naming Your Filters

As you continue to create filters for the GCS and other views, be sure to assign appropriate names so that anyone can quickly understand what they are being used for. A nondescriptive filter name is really good for nothing. Naming a filter that removes hangers from a GCS "Hangers" indicates that it is a selection of hangers, when in actuality its purpose is to remove all hangers from the GCS. A name like "Hangers Omitted From GCS" is much more descriptive.

Adding a parameter called Column Usage will allow you to give each structural column a property explaining its purpose in your project. Maybe the column is being used as a pier, or a hanger, or even a smaller post and you don't want it to show up in the GCS because it will be scheduled with a different method. You can apply such values as Pier, Hanger, or Post, and use filters to prevent those columns from displaying in your GCS.

As you can see, the possibilities of using filters in the GCS are endless. They can give you the freedom you need to produce great-looking column schedules. You can take advantage of filters in any other view throughout a project using the same procedures and methods. When used properly, filters are a great way to add flexibility when producing your documentation.

The Bottom Line

Work with the basic structural column family template Understanding the differences between the various templates for a structural column will help you ensure that your structural columns behave properly when placed into your project.

> **Master It** What setting in a structural column family gives it the characteristics that help Revit Structure determine connection and attachment properties and whether the columns display in the Graphical Column Schedule?

Place structural columns in your project Knowing all the methods available for placing structural columns in your project and knowing when to use each one will help you place columns quickly and accurately into your project.

> **Master It** What are three methods you can use to place structural columns in your project, and how can you quickly place them onto your upper levels?

Attach structural columns to other structural components To help maintain the top and bottom of a column's location and relationship to other elements, they can use different cut styles and justifications while being attached to various elements.

> **Master It** What are four types of elements that structural columns can be attached to?

Employ the methods of placing a sloped column You can use several methods to place sloped columns into your project even though Revit Structure does not offer current families and programming that fully work with sloped columns.

> **Master It** What are some of the methods used to create sloped columns?

Document your model with the Graphical Column Schedule The Graphical Column Schedule can be generated automatically by Revit Structure and modified to prevent columns from showing up that you do not want displayed.

> **Master It** What are three methods used to remove structural columns from the Graphical Column Schedule?

Chapter 5

Floor Slabs and Roof Decks

Floors and roofs are two of the main elements for any building project, and you must contend with them when you are modeling a structure. The various floor construction types that you will encounter on a regular basis include concrete slabs-on-grade, and supported decks constructed with concrete and metal deck, wood, or formed concrete. Along with modeling the basic slab or deck profile, you will also be required to add slab edges, depressions, ramped floors, and openings to your project in order to accurately develop the design.

Although the techniques seem similar, roof modeling is quite different from floor modeling because you will be spending a lot of time and effort sloping the roof for drainage purposes. Roofs come in many forms, from a residential wood Cape Cod–style roof to a commercial roof that warps from ridges to drains. A roof is one of the most difficult elements to model in your projects; therefore, the tools and workflow required to create and edit roofs must be up to the task. It is essential to gain a solid understanding of how to model these difficult elements to ensure the integrity of your model is maintained.

Revit Structure has many excellent tools to add to your modeling tool chest that help in the creation of these floor and roof entities. In this chapter, you will see those tools in action.

Of course, the slabs will have reinforcing bar or mesh in them, but you will learn how that works later in the book in Chapter 12.

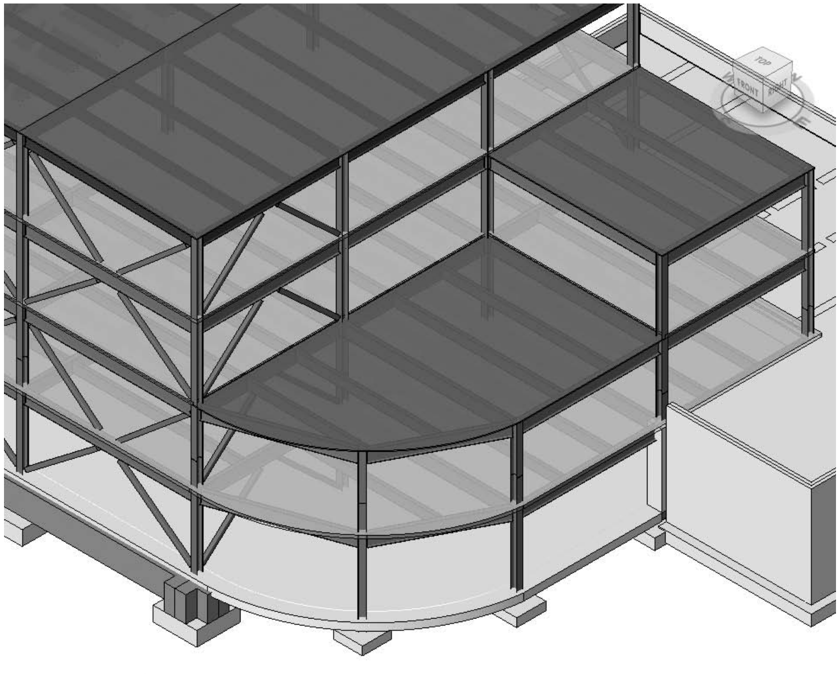
In this chapter you will learn to:

- Create a slab-on-grade with dropped slab edges
- Work with floor decks
- Work with warped roof decks
- Create openings and depressions in your floors and roofs

Floor Slabs and Decks

The first part of this chapter will deal with slabs-on-grade as well as elevated slabs and decks as illustrated below. An understanding of their basic properties and the tools used in their creation is an essential step to mastering Revit Structure. In this section you will have a chance to work with these elements in order to create various slab conditions. You will learn how floor types are made within the floor system family, and how you can modify them to satisfy the many conditions

that you will be required to model in the virtual environment. To that end, you will examine many forms of supported deck as well as slabs-on-grade. First let's take a look at the basic properties of slabs.



Slab Properties

You can find the Slab command on the Design bar and on the Modeling menu. When you click the Slab button, you will enable Sketch mode. Sketch mode isolates you from the rest of the model as you are working on the slab so that you can create its outline without disturbing other elements. Sketch mode is used not only for making slabs but in many other functions as well. You should have a thorough understanding of it if you want to use Revit Structure effectively.

As discussed in earlier chapters, most elements that you model have instance and type parameters (see Figure 5.1). The type parameters are ones that apply to every occurrence of an element in your project. The instance parameters are ones that occur for a particular insertion of an object. In the case of slabs, an example of a type parameter is the thickness of the slab. An instance parameter example is the height from the level to which the slab is attached.

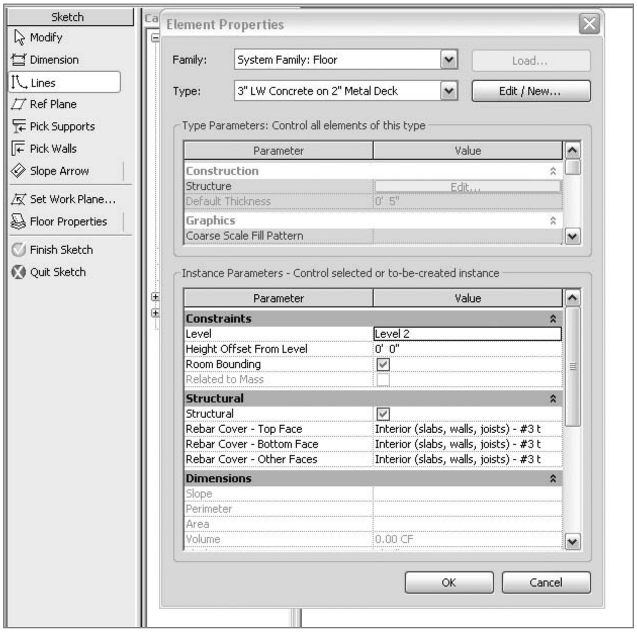


FIGURE 5.1
The Element Properties dialog box for a floor

Once in Sketch mode, you create the slab by using any one of the available suboptions:

Line Use this option to sketch the boundary of the slab.

Pick Supports Use this option to select beams or girders.

Pick Walls Use this option to select walls that support the slab.

Reference Planes Use this option to attach your slab to a reference plane.

Slope Arrows This option allows you to set slab heights for two edges of the slab relative to the level or reference plane where you are working.

Using one of those methods, you draw the boundary of the slab. The slab boundary lines must be a closed sequence of lines or arcs, so you must be sure that your elements are trimmed and connected to each other. Otherwise, Revit Structure will not allow you to complete the sketch.

Once the boundary has been established, you will assign a slab type to the element (see Figure 5.2). Some slab types are available as defaults; some you will need to create yourself in the properties dialog box (see Figure 5.3).

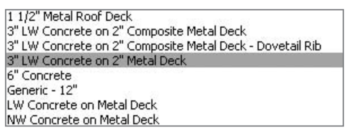


FIGURE 5.2
Default slab types

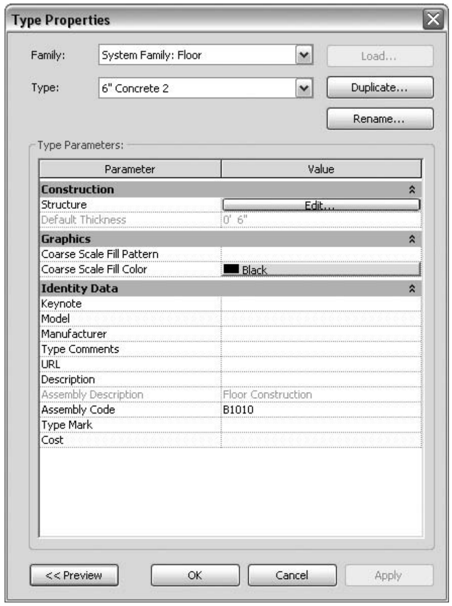


FIGURE 5.3 Type Properties dialog box for a floor element

Within the slab family, you can create many types of floors. Let's now explore the workflow you will use to produce these elements in various forms, starting with a slab-on-grade.

Slabs-on-Grade

The most basic slab you can make is a simple slab poured against the ground, known as a slab-on-grade. You may have gotten out the hoe and wheelbarrow and mixed some concrete in your own backyard at some point to make a new slab for parking, or a basketball court, or a walkway perhaps. You began by laying out the boundary with boards or digging and letting the earth be your form, adding in some reinforcing if needed, and then poured the concrete. You may also have had to block out certain portions to create openings or depressions. So that is exactly what you have to do here: create a slab boundary, and then apply a certain type of slab to it.

ADDING A SLAB TO YOUR PROJECT

Let's examine the method that you will use to create a slab-on-grade within Revit Structure:

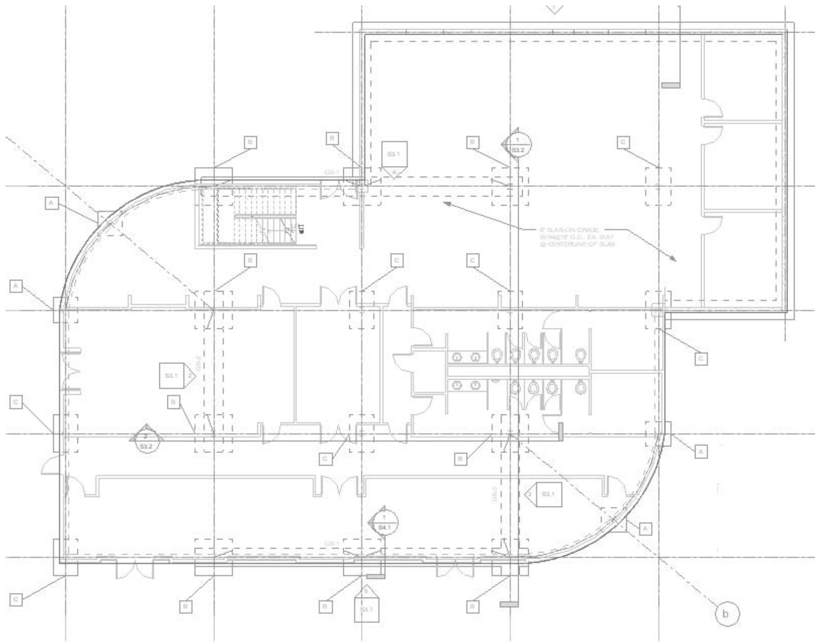
1. Go to the Ground level of your project.
2. Click the Modeling menu on the Design bar.
3. Click the Slab button, which puts you into Sketch mode.

To draw the slab boundary, follow these steps:

1. Click Lines on the Sketch menu.

2. On the Options bar, you can choose to either draw the lines, or to select an existing line that will be duplicated in the same location.
3. In the Element Properties dialog box, choose an appropriate slab type, like a 6″ concrete slab.
4. Click OK to exit the Element Properties dialog box.
5. Complete the slab boundary (see Figure 5.4). If necessary, use the Trim command to ensure that all lines are connected.

FIGURE 5.4
Sketching the slab-on-grade



The slab lines can be locked to either the framing supporting below or to the architectural stud line in order for the slab lines to adapt automatically to a change in size. Then click the Finish Sketch button, and the slab is complete.

You need to be careful that you don't overconstrain the model because the performance could suffer. Sometimes you might not want to lock your sketch lines to their boundary for this reason. It takes some time and experimentation to get a feel for those limits.

After the slab is complete, Revit Structure automatically adds a notation for the span extents and direction to the view.

That was easy enough to do, but what if you need a 4″ slab for your project, and you want a 2″ layer of sand below it? In that case you will create a new slab type by duplicating the existing 6″ concrete slab and then adapting its properties. You will also add a new layer to the 4″ slab. The following section shows how the basic procedure works.

Creating a New Slab Type

To create the basic slab type, follow these steps:

1. As you did in the previous steps, go into Sketch mode and click Floor Properties.
2. In the Element Properties dialog box, click Edit/New to open the Type Properties dialog box, where you will create the new slab type.
3. Select the existing slab type that is closest in structure to the one you wish to create.
4. Click the Duplicate key.
5. The Name dialog box appears. Give the slab a name, such as **4" `concrete slab`**, then click OK.

To add layers to the new slab type, follow these steps:

1. In the Type Properties dialog box, click the Edit… button in the Structure section
2. The Edit Assembly box appears (see Figure 5.5). This dialog box allows you to alter the structure of the new slab.
3. The slab assembly is made of layers, which are inside or outside of a defined core boundary. You add layers to the slab definition by clicking the Insert button at the bottom of the dialog box.
4. Once a layer is inserted, you can then move it up or down relative to the other layers by clicking the Up or Down button at the bottom of the dialog box (see Figure 5.6).
5. When you click into the Material field of each Function click the small button on the right side of the box. That will display the Materials dialog box where you can change the material options.
6. In the resulting Materials dialog box, you can choose various materials, alter the existing material, or create new materials.
7. In the Thickness field, specify **4**".
8. Finally, check the Variable check box if the slab depth has a variable fill.

The variable fill allows you to vary the thickness of the slab by raising or lowering its top. The bottom of the slab remains constant.

9. Click OK in the remaining dialog boxes.
10. On the Design bar click Finish Sketch to complete the slab.

Editing the Slab Element

Editing the slab once it has been created is not difficult. Just follow these steps:

1. To change the slab type, highlight the slab and select a new type in the Project Browser.
2. To edit the boundary of the slab, highlight the existing slab.

3. Click Edit on the Options bar, which puts you back in Sketch mode.
4. Edit the slab lines as needed with the Sketch tools.
5. Click Finish Sketch.

FIGURE 5.5
The Edit Assembly dialog box

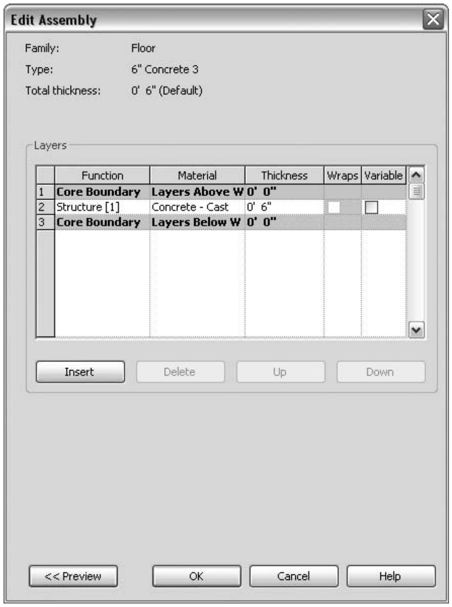
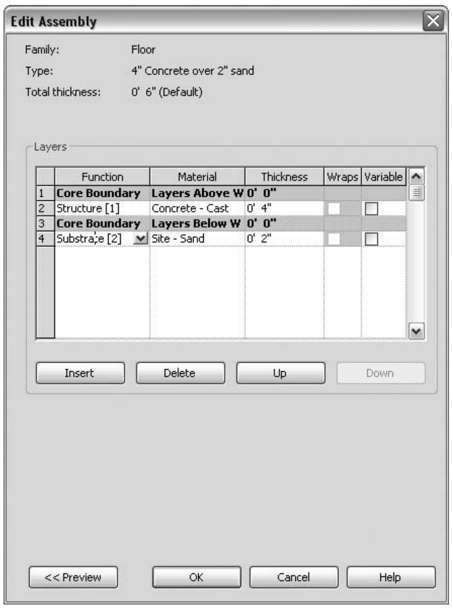


FIGURE 5.6
Adding a layer to the slab structure



Exercise: Creating a Slab-on-Grade

In this exercise, you will create a basic concrete slab-on-grade with a vapor barrier below it. First, you sketch a basic rectangular shape. Then, you create a new slab type consisting of 4″ concrete with 2″ sand beneath it for the shape.

To sketch the basic slab shape:

1. Open a new project in Revit Structure.
2. In the Project Browser, double-click on Level 1.
3. On the Design bar, click Modeling, then click Slab, which enables Sketch mode.
4. Click Lines on the Sketch menu.
5. On the Options bar, click the Rectangle button.
6. Draw a rectangle of any size for the slab boundary.
7. Click the Floor Properties button on the Design bar.

Now let's create the new slab type:

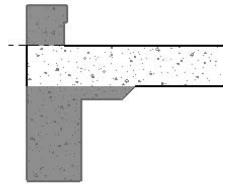
1. In the Type pull-down, select **6″ Concrete**.
2. Click Edit/New, then in the Type Properties dialog box, choose Duplicate.
3. In the Name dialog box, type **4″ concrete over 2″sand**, and click OK.
4. In the Structure field, click Edit.
5. In the Edit Assembly dialog box, change the thickness to **4″**.
6. Click the Insert button to create a new layer below the concrete.
7. Move the new layer below the core boundary layer of concrete.
8. Click the Function field and change it to Substrate 2.
9. Click the Material field and change the material to Site-Sand. Click OK and then click OK again to exit.
10. Click the Thickness field and change the value to **2″**.
11. Click OK in the remaining dialog boxes.
12. Click Finish Sketch to complete your slab.

Leave your model open for the next exercise.

Slab Edges at the Ground Level

Creating the basic slab-on-grade is not too difficult. But each slab is going to have an edge condition that will also have to be modeled. For the basic slab-on-grade that we have been working on so far, there will most likely be a thickened slab 12″ to 18″ in depth around the perimeter of the slab, with a bearing surface of 8″ to 12″ (see Figure 5.7). Your next task while building your model may well be to add a slab turndown edge to your slab element, as shown in Figure 5.7.

FIGURE 5.7
Slab edge with curb attached to the slab



The Slab Edge function is located on the Modeling menu. The shape of the slab edge is controlled through the use of slab edge profile families. These families are parametrically driven, giving the profile family the ability to flex to different sizes of the same basic shape. There are limited types loaded into the default template, so you will most likely need to add new profile types, which you will learn to do later in this chapter. For now, let's explore the basic procedure for adding a slab edge type to your project.

On the Modeling menu, click Slab Edge, and then select the slab edge type you want to use from the Type Selector. To add the slab edge to the slab, just begin clicking on the edges and make your way around the slab. The slab edges can be grouped together if different types are required at different parts of the slab. To start a new set of edges, simply click the Next button on the Options bar. By making groups of slab edges, you can better control and define their behavior, especially if you have multiple types. When viewed in section, the slab edges will "clean up," or appear monolithic with the slab as long as the two elements have the same material.

Now you've completed your basic slab edge. But during the period that you are designing your structure, you will undoubtedly need to revise the slab and its edges several times. You might want to eliminate one segment or more that you have added. How do you do that? First, you highlight a slab edge segment that you want to change. On the Options bar, click Add or Remove Segment, then select the segment you want to remove or an edge where you want to add a slab edge.

After placement, the slab edges can be offset vertically or horizontally from the slab element. To offset the slab edge, simply highlight it, and then click the Element Properties button to access the dialog box. There you can adjust the position of the slab edge by adding values for the vertical and/or horizontal Profile Offset parameter.

You can also flip the slab edge profile 180 degrees in plan view. First, highlight the segment to flip. Then click the double arrow control that appears next to the slab edge and that will mirror it about the edge of slab.

In practice, you will contend with many different types of slab edges, and it can become difficult to keep them accurately modeled and displayed. It tends to be a time-consuming activity. But in order to display the model accurately, it is a necessary concern. Let's take a look at the profile creation process a little closer and see how a new profile type can be made and loaded into your project.

MAKING A NEW SLAB EDGE TYPE

If you have to make a new slab edge type in your project, most likely you will also have to add a new type to the slab edge profile family because they work together. Let's examine the general procedure.

First, add a new type to the profile family:

1. On the menu bar, click File ➢ Open.

You will need to open the external profiles family for slab edges.

2. In the Look In window on the left side of the Open dialog box, click Imperial Library.
3. Double-click the Profiles folder then the Structural folder and finally select the `Slab Edge-Thickened.rfa` file (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`). This will open the profile family.
4. Click the Family Type button on the Design bar.
5. Click the New button and give the new profile type a name, such as **12″ wide x 18″ dp**.
6. Change the width and thickness values in the Dimensions field of the dialog box for the new size.
7. Click the Apply button at the bottom in order to "flex" the model and test the new shape.

"Flexing" the family type is an important thing to do as you develop your families. It's like a test-drive that ensures that the shape is accurately changing to your satisfaction.

8. Click OK in the remaining dialog boxes.
9. On the menu bar, click the Save button to save the new profile type to the file.
10. On the Design bar, click Load into Projects.
11. When prompted, click OK to overwrite the existing profile definition.

With the profile created, let's now incorporate it into the slab edge type:

1. Highlight the slab edge.
2. Click the Element Properties button (next to the Type Selector) in order to create a new profile type in your project.
3. Click Edit/New and give the new slab edge a name.
4. In the Profile field, select from the pull-down list the slab edge profile you created earlier.
5. Click the button on the right side of the Materials field to open the Materials dialog box.
6. Select Cast-in-Place Concrete. Then exit the remaining dialog boxes.

You want the slab edge material to be the same as the slab material so that when you cut a section the final display of the slab and edge appears monolithic.

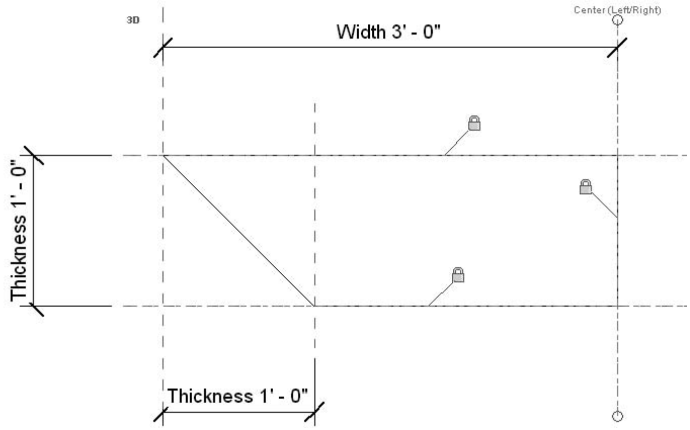
Your new slab edge type is now complete.

Of course, that was a simple slab edge shape in which you added a new type to an existing slab edge family. You can also start your own profile family file, and create any shape you might need. You can then add constraints to the basic shape to make it become a parametric family whose shape can flex to different sizes.

The parametric nature of Revit Structure families means you can create and manage literally thousands of modeling shapes in a well-coordinated, easy-to-understand fashion.

Figure 5.8 shows the profile family for the edge of slab. It could be just a single four-sided shape for use as a profile. You could then create many different files, each with a similar four-sided shape but with different widths and thicknesses, such as 12 × 36, 10 × 20, 18 × 36, all in a library of edge types. AutoCAD works that way. That approach is nonparametric. To have one profile family, representing similar shapes, with the ability to flex the shape easily into new types is the parametric approach. Revit Structure works that way.

FIGURE 5.8
Parametric constraints in the slab edge family file locks the shape lines to reference planes.



THE POWER OF PARAMETRICS

One of the real powers of Revit lies in the parametric nature of its model and annotation families. You should learn to create parameters in your own families that allow the family elements to flex into new shapes. But if you get overly frustrated in your first attempt to do this, then stick with creating the single shape family. Later you can evolve to the higher-level creation process.

CONVERTING A SINGLE SHAPE TO A PARAMETRIC SHAPE

Next, let's see how to convert a simple single shape (see Figure 5.9) to a parametric one. The general process gives you an insight into how to adapt and extend your modeled shapes into more generalized families:

1. Draw reference planes on the planes where a surface will be placed.
2. Draw lines along the reference lines to create the shape of the slab edge.
3. Lock the lines to the reference plane as you are drawing them. You can also lock ends of lines to reference planes.
4. Create dimension strings between the reference planes.
5. Highlight one of the dimensions, right-click, and choose Edit Label. In the resulting pull down box, click Add a Parameter, or choose to edit an existing one.
6. In the resulting dialog box create or edit the parameter label.

7. Attach flexible parametric labels to the dimensions, which act either as type or instance parameters within the family.
8. Create a new type.
9. Flex the model to make sure it is working properly.

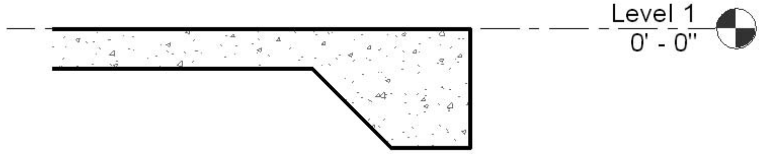


FIGURE 5.9
Finished edge of slab

EXERCISE: CREATE A SLAB EDGE TYPE AND ADD IT TO YOUR SLAB

In this exercise, you will create a new type from the loaded slab edge profiles, and then you will add it to the slab you made in the previous exercise. When you complete the exercise, make a section through the slab and check to see if the slab edge is displaying correctly.

First, have a slab available in an open model from the previous exercise, or create one in a new model. You can now create a new slab edge type in the profile family:

1. On the menu bar, click File ➢ Open.
2. In the "Look in" section at the left of the Open dialog box, click the Imperial Library icon.
3. Double-click the Profiles folder then the Structural folder and finally select `Slab Edge - Thickened.rfa`.
4. On the Design bar, click Family Types.
5. Click New, and name the new profile type **30″x20″**
6. Enter the values **30″** for the width and **20″** for the thickness.
7. Click the Apply button at the bottom of the dialog box to apply the revised dimensions to the shape.
8. Click OK to exit the Family Types dialog box.
9. On the toolbar, click the Save button.
10. Append your initials to the filename and click Save.
11. On the Design bar, click the Load into Project button.

That takes you back to the original file, where you will now add the slab edge by doing the following:

1. Select Slab Edge from the Modeling tab on the Design bar.
2. On the Options bar, click the Properties button (to the right of the Type Selector).
3. Click Edit/New, then click Duplicate.
4. Assign the new slab edge type the name: **Thickened 30″ x 20″**. Then click OK.
5. In the Type Profile field, select the Slab Edge - Thickened 30″x 20″profile type that you created.
6. Click in the Type Material field to display the Materials dialog box.

7. Select Concrete – Cast-in-Place Concrete, and click OK in all remaining dialog boxes.
8. Click each edge of the slab successively to form a chain of edges around your slab.
9. On the View Control bar, change the view to Hidden Line to show the edge hidden below correctly.
10. Highlight the slab edge you just created.
11. On the Options bar, click the Add or Remove Segments button.
12. Click on the right and left edges of the slab to remove those segments.

Floor Deck Creation

The next important aspect we will explore is working with floor decks in Revit Structure, with a focus on the nature of multilayered decks.

Here are some examples of floor decks you might encounter in projects:

- A steel project might have a floor deck consisting of 3¼″ lightweight concrete over 3″ × 18″ gauge metal deck.
- A wood floor deck on the second floor of an elementary school could be 2″ of lightweight concrete fill over a ¾″ layer of wood sheathing.
- A concrete flat slab over a basement could have second fill topping at the street level for drainage purposes.

The process of creating types of floor deck (see Figure 5.10), and their placement, is the same as for the slab-on-grade, except for several important aspects, most notably creating composite decks. The metal deck family has several default types, as shown earlier in the chapter. You learned how to add layers when you added a vapor membrane below the slab-on-grade. To work with metal deck types, you must understand how they can be created.

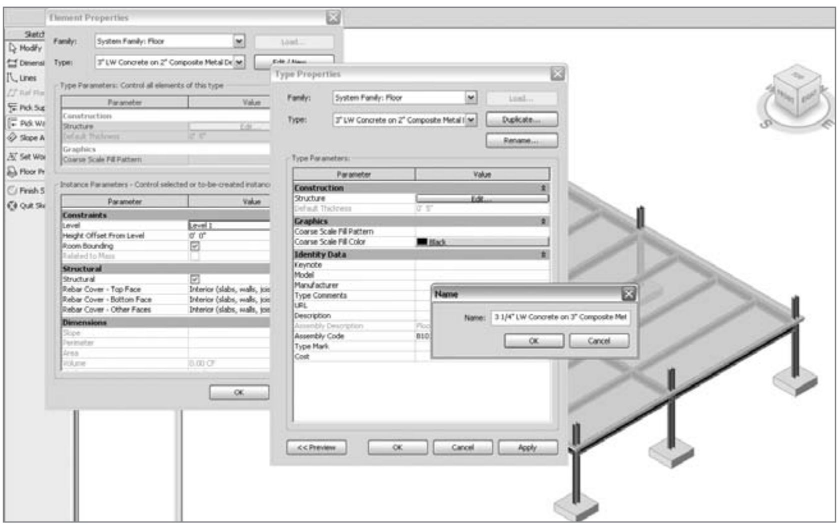


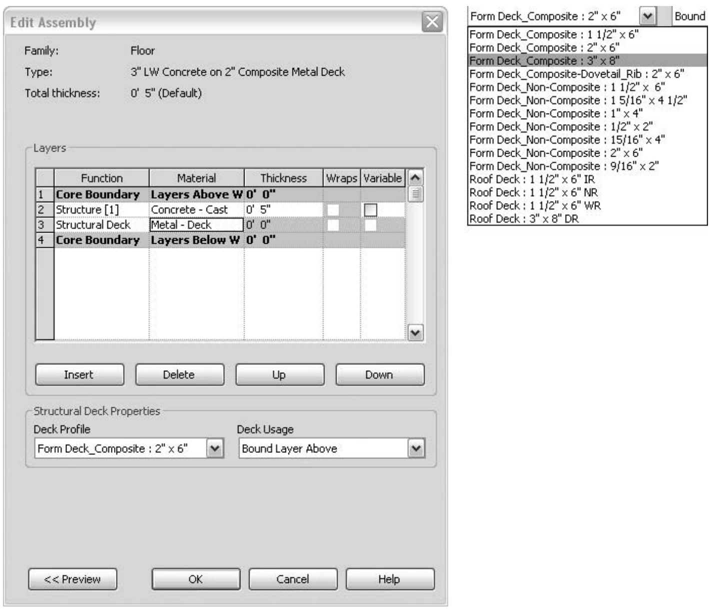
FIGURE 5.10 Giving the new deck type a name

Making a New Composite Deck Type

The process of making steel and concrete composite decks is much the same as the basic slab-on-grade except the deck is multilayered. It is important to create a composite deck type in such a way that it can be cut in section and will accurately show the fluted metal deck filled with concrete. That helps make your initial section much more complete so that finishing it will require much less effort

1. On the Modeling menu, click Slab and then choose Floor Properties.
2. In the Element Properties box, click the Edit/New button.
3. Click the Duplicate button, and then give the new deck a name.
4. In the Structure parameter, click the Structure button.
5. Within the core boundary, insert a layer of concrete.
6. Insert a layer of metal deck.
7. Click on Metal – Deck in the Function column of the Edit Assembly dialog box. Choose either Metal or Structural Metal Deck.
8. If you choose Structural Metal Deck, you can choose a deck profile from the Structural Deck Properties area (see Figure 5.11) at the bottom of the dialog box. Those profiles can be edited in the Metal Deck profile family.
9. From the Deck Usage dropdown, choose Bound Layer Above.
10. If the structural deck profile does not exist, you can create one by editing the deck profile family and adding new types, similar to the way you did with the slab edge profile.

Figure 5.11
Defining the metal deck structure in the Edit Assembly dialog box



Adding a Cantilever to a Floor Deck

Edge conditions are a major consideration for floor decks. Typically the metal deck stops at the support girder and then only concrete extends out to the inside face of the stud supported by an angle or a bent plate (see Figures 5.12 and 5.13). So in order to accurately represent the edge condition in your model, the deck family has to accommodate the discontinuity of the metal deck. The procedure is not difficult to accomplish. While in Sketch mode, highlight one of the boundary lines. Then on the Options bar, set the concrete or steel cantilever distance (see Figure 5.14) as required for your project. You will notice that a positive value makes the extension go to the inside of the support line, and a negative value makes the extension go to the outside of the support line.

FIGURE 5.12
The edge of the deck is parallel to the deck flutes.

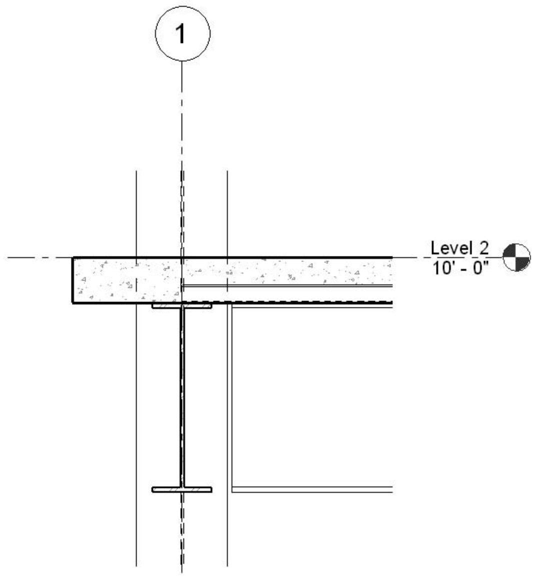


FIGURE 5.13
The edge of the deck is perpendicular to the deck flutes.

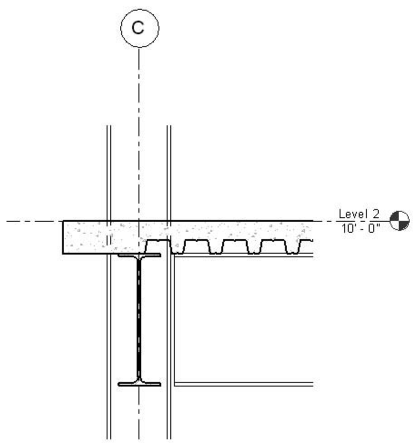
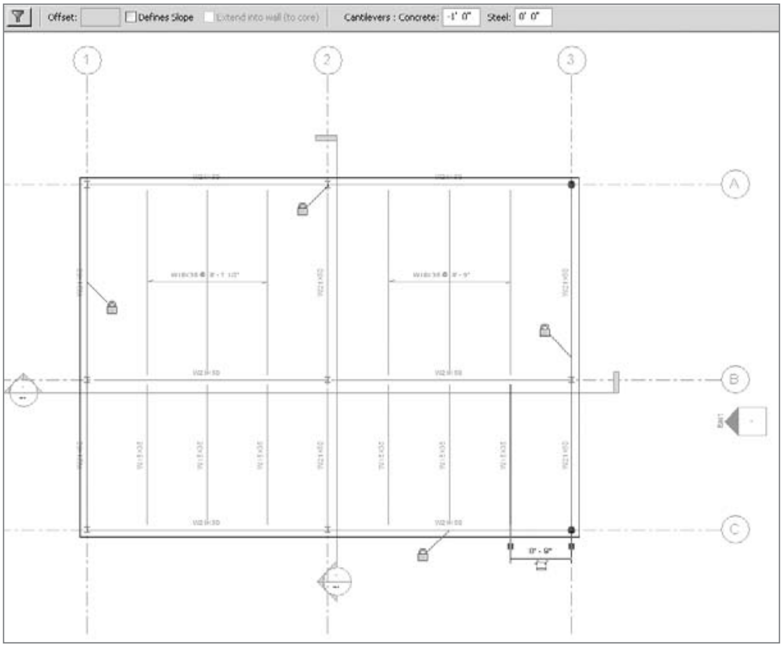


Figure 5.14
Adding deck cantilevers in plan view



The span direction is another important element to consider. When the deck is first created, a span direction indicator is added. The half arrows shown in one direction indicate the strong direction of the deck. You can also add the span indicator at any time by selecting the Drafting tab on the Design bar and clicking Span Direction. If you highlight the span direction indicator that is already placed and rotate it, you can change the direction of the deck. This will be essential to show the deck correctly in your section views.

Other Examples of Floor Type Construction

By now you might be asking, "What about different types of concrete formed structural slabs with decks, such as PT slabs, pan joist waffle slabs, and double-T slabs?" For those slabs, refer to Chapter 7 (on framing) for a full discussion. Since slab and framing are integrated, we chose that chapter to discuss the creation of those types of elements.

Roofs

Now that you have started to get a feel for creating floors, you will find that roofs are created in much the same fashion. The big difference lies in the many varieties of wood, steel, and concrete roof shapes that you must be prepared to accommodate in your project. It could be a residential gable roof, salt box roof, a flat commercial-style roof deck, or perhaps even a freeform, wavelike shape.

To accomplish this task, you will use special tools to help create and slope your roof elements. You will then also be able to combine different roof shapes and slopes into one roof system. In Chapter 7, you will learn how to best frame your roof for these different roof conditions. Of paramount concern in constructing your roof system is the ability to edit the roof sloping and the framing that supports it as easily as possible as the design progresses.

As you will see in Chapter 7, a sloped and warped roof will necessitate that your beam and girder system supporting the diaphragm “flex” automatically when you edit the diaphragm. Since each beam will most likely have a different elevation at its end, editing a complex roof shape can be time consuming. Therefore, you will need to give special consideration to your approach.

One big issue that often crops up in many projects concerns the interaction with the architectural designers as you develop the roof for your model. If the architects still use 2D techniques, they will normally not be too concerned in the design development stage about the sloping of the roof. But for the person modeling, that can mean trouble. Best practice is to try to work with the architect to get an idea of ridge and drain locations that you can then readily edit later in the project. To create your roof element, you will need enough information about the sloping so that you can at least get a semblance of the roof system put into place. If you simply begin by creating a flat roof, you will undoubtedly be forced later to scrap it and start over. Also, keep in mind that the integrity of the model is compromised showing just a flat roof.

Now you will have an opportunity to take a close look at the techniques used to model different roof conditions. The two main areas we will focus on are the geometry of the roof diaphragm and the creation of various roof types to apply to that geometry.

Roof Deck Properties

The roof family, like other families in Revit Structure, is populated with different types. These types are basically created in the same way as floor types were created earlier in this chapter. The two basic families of roof types are Basic and Sloped Glazing. We will concentrate on the Basic type. Sloped Glazing roof types are appropriate for creating a greenhouse or a skylight structure, and work similar to curtain walls, except they can slope.

Let’s now examine the process of creating a roof deck. You find the Roof command not on the Modeling menu, but on the Architectural menu. There are four different ways to create a roof deck with this command, and we will concentrate on two: Create by Footprint and Create by Extrusion.

> **The Architectural Tab**
>
> **You may not be able to find the Architectural tab on the Design bar. By default, it is not active in Revit Structure. To show the tab, right-click on the Design bar and select Architectural from the list.**

Using the Create by Footprint Option

Now let’s see how to create a roof deck using the Create by Footprint options:

1. Go to your roof level view, then on the Design bar, click the Architectural tab, select Roof, and then select Roof by Footprint.
2. On the Design bar, click Roof Properties, which displays the Element Properties dialog box.

3. Click Edit/New, and in the resulting Type Properties dialog box, click Duplicate.
4. Give the new roof type an appropriate name.
5. In the Type Properties dialog box, under Construction, click the Edit button in the Structure field.
6. In the Edit Assembly dialog box, you will create the different layers of the deck, such as a metal deck layer, and a fill over that.
7. A typical roof assembly fill might be a lightweight concrete fill, wood sheathing, or solid insulation. These components of the roof assembly will be within the core layer. Create an assembly fill layer by clicking the Insert button and assigning to it material, and depth definitions.
8. Use the Up and Down buttons to move the layers as needed to their correct location within the roof element.
9. Create other nonstructural layers, such as roof membranes, outside of the core boundary.

Roof Crickets

Adding Roof crickets to your roof is difficult to accomplish and usually are left to the architect to model since it is not really a structural concern. You, on the other hand, will want to concentrate on the underside bearing surface of the deck in order to prepare for the addition of support framing.

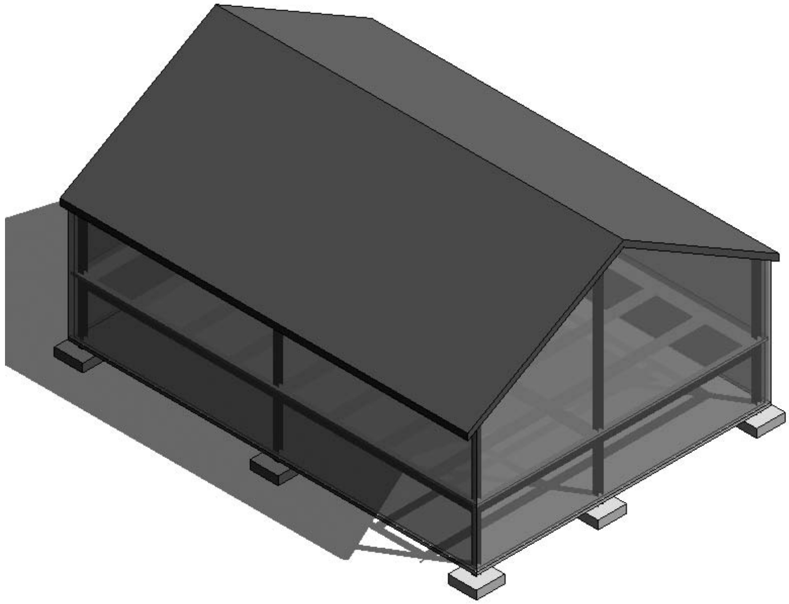
10. You must configure two basic characteristics when creating the deck layers: the material and depths of each layer. Click in the Materials box of a layer to display the Materials dialog box. For this example, choose Default Roof. You can duplicate and then edit materials as needed.
11. Click the Physical tab to see the analytical properties of the layers.
12. Transparencies are often applied to the materials so that you can see through the roof deck and see the framing below. Set the Transparency value to 10%.
13. Click in the Thickness field of the layer and set the thickness to **2**".
14. Check the Variable box to the right of each layer to free up that layer so that you can vary its depth.
15. The bottom of these layers will remain constant, while the tops can vary. First, set the minimum thickness and then check the Variable box to have thicker portions than the minimum setting.
16. Click OK in all remaining dialog boxes.

You have successfully created a new roof deck type. Now it is time to move on to sketching the actual deck geometry.

Sketching Roof Geometry for Your Project

As stated earlier, sketching roof geometry is one of the most challenging things you will do when modeling with Revit Structure. Let's start by looking at some of the easier forms and work our way up to ones that are more difficult. First, let's create a sloped residential roof (see Figure 5.15).

Figure 5.15
A simple form of residential roofing



The basic residential-style roof (as shown in Figure 5.15) is not difficult to model. In general, just do the following:

1. On the Design bar, click the Architectural tab, choose Roof, and select Roof by Footprint.
2. You will enter Sketch mode, where you can now draw the outline of the roof.
3. Click Lines on the Sketch menu.
4. On the Options bar, the basic Sketch tools are available so you can draw or select the outline, or "footprint," you want.
5. Once you create the outline, highlight the lines that will be at the top or base of the sloped surfaces—in this case, the line on each long side of the roof.
6. On the Options bar, check the Defines Slope box.
7. On the highlighted lines, you will see an angle symbol next to the line that represents the slope value (see Figure 5.16). Click in the box and enter the slope value.
8. Click Finish Sketch to exit, and Revit Structure creates the roof.

Formatting the Slope Value

The format of the slope value by default is expressed in degrees, such as 30°. It can also be expressed in several other ways, such as inches per foot, which in many cases is how the architect will specify it. To change the display to inches per foot, click Settings ➢ Project Units ➢ Slope, then change the Units format.

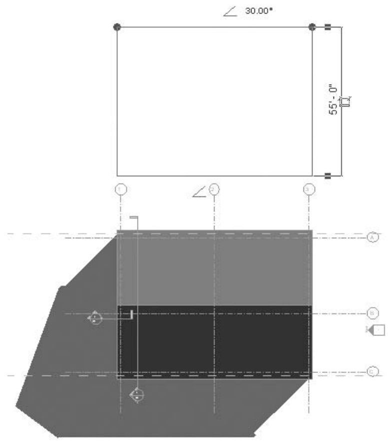


Figure 5.16
Sketching a basic roof and adding slope angles to the base lines

Figure 5.17 shows a complicated hipped roof that slopes in four directions and adds dormers on each side.

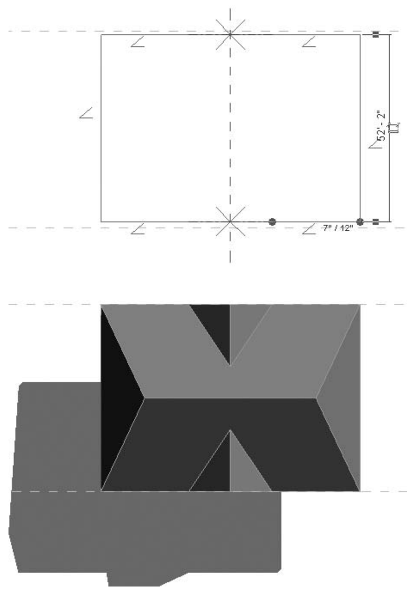
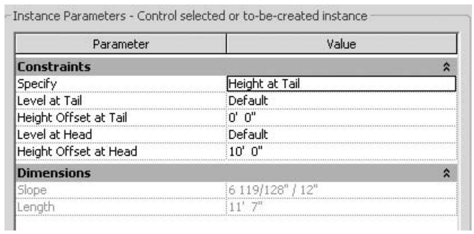


Figure 5.17
A basic roof with hips and dormers

Creating a Basic Roof with Hips and Dormers

The basic way to create hips and dormers involves these steps:

1. Add lines with slope angles on all four sides, then adjust each slope angle accordingly.
2. Use slope arrows to create the dormer portions. Slope arrows indicate the direction and extent of a sloping deck line, such as a dormer.
3. On the top and bottom lines of the roof footprint, create four lines, each representing the basic roof or dormer horizontal length.
4. Click Slope Arrow on the Design bar.
5. On the dormer lines, add the slope arrows with each arrow pointing toward the middle.
6. Highlight a slope arrow, then click the Properties button on the Options bar. Repeat as necessary for each slope arrow.
7. Adjust the properties to fit your dormer configuration (see Figure 5.18).



Instance Parameters - Control selected or to-be-created instance

Parameter	Value
Constraints	
Specify	Height at Tail
Level at Tail	Default
Height Offset at Tail	0' 0"
Level at Head	Default
Height Offset at Head	10' 0"
Dimensions	
Slope	6 119/128" / 12"
Length	11' 7"

Figure 5.18
Slope arrow properties

Creating a Roof with a Cutoff Portion

Another common roof shape you might encounter involves cutting out a portion of the roof to create a cutoff condition (see Figure 5.19). The general process for this example is the same as for the roof that you just created. Once you complete the basic shape, highlight it and then click the Element Properties tab next to the Type Selector. In the Element Properties dialog box, you can add a value to the Cutoff Level parameter to establish a point above the base of the roof where you would like it to be cut.

Getting the idea of how this works yet? Play around with these various options until you feel a little more comfortable with how this all functions. Using these various roof tools, you can create a complicated roof diaphragm like that in Figure 5.20 fairly easily.

Play Around with the Options

One thing that becomes clear when you start modeling with Revit Structure is that it is a satisfying way to work and is actually fun. That can be quite disconcerting to those who never equate work with having fun. "I'm working, I'm not supposed to be having fun!" some will say. But all kidding aside, working in three dimensions rather than two activates different parts of the brain and awakens a different and deeper understanding of the building that you are creating.

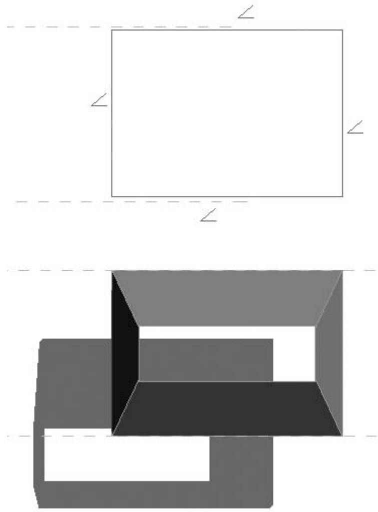


FIGURE 5.19
A cutoff roof element

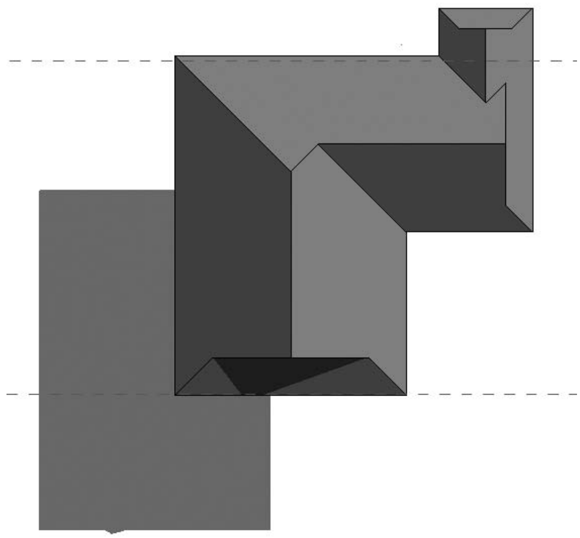


FIGURE 5.20
A complicated residential roof with joined multisloped portions

By sketching the footprint and applying slope angles to the edges, you can create some distinctive roof shapes (see Figure 5.21).

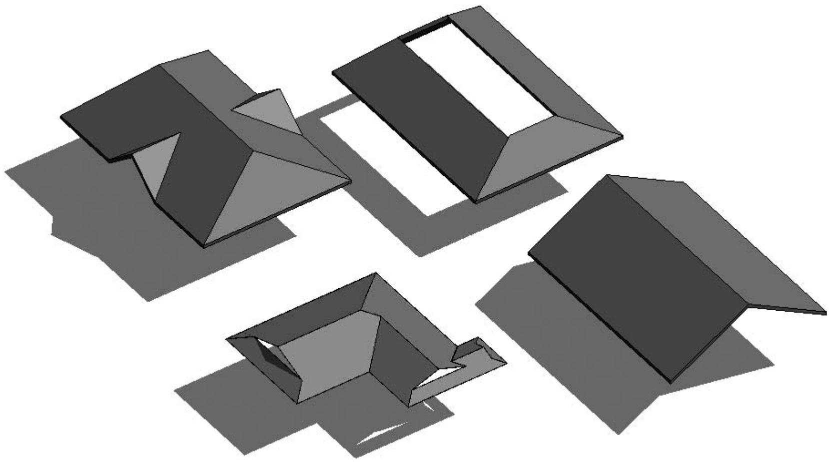


FIGURE 5.21
View of some typical roof types

Another way to create a roof diaphragm is by using the Roof by Extrusion option. This option is a good one to use if you have a fairly straight multiple-sloped roof system with areas that need to be cut out.

ROOF BY EXTRUSION METHOD

The roof by extrusion method uses a profile of the roof shape that you want to create, which is then extruded over the length of the new roof segment.

1. In plan view, draw a grid. This will be used as the work plane to sketch the profile in a section view of the roof extrusion.
2. On the Design bar, click the Architectural tab, choose Roof, and select Roof by Extrusion.
3. Choose grid you set up as the work plane which you will use to draw the profile in section view.
4. Select the level and the offset to that level to which you will attach the roof.
5. In the section view, draw a line that represents the roof shape.
6. Click the Properties box on the Design bar and add values for Extrusion Start and Extrusion Finish.

Extrusion Start and Finish Values

You do not need to be exact in establishing the Extrusion Start and Finish values, as you will soon see. Each value is relative to the reference plane or grid on which you chose to draw the profile. Enter one value as a negative and one as a positive value, say –5´-0˝ and 5´-0˝.

7. After you exit the Properties box, go to the roof plan view.
8. Now you can stretch out the roof in either direction relative to the current reference plane (see Figure 5.22).

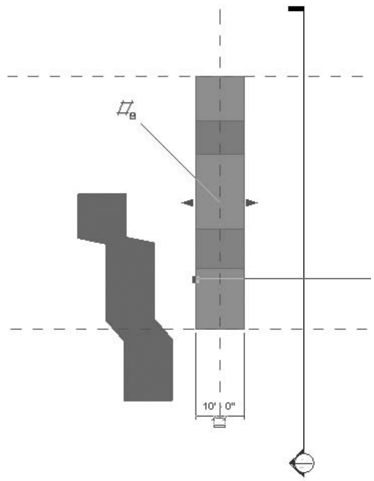


Figure 5.22
The roof by extrusion method: stretching the roof extents by clicking and dragging the blue triangles

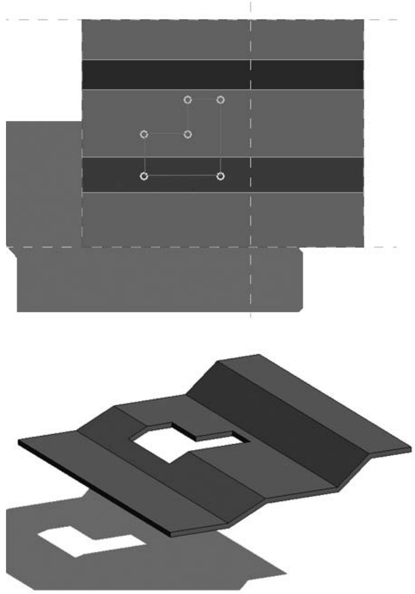
After you have finished the basic roof by extrusion, you can add openings for light shafts or other purposes such as mechanical openings as illustrated below.

To add openings you create solid voids that cut out portions of the solid extrusion you just created (see Figure 5.23).

To cut an opening in the roof extrusion:

1. In plan view, highlight the roof element.
2. On the Options bar, click Cut Plan Profile, thus enabling Sketch mode.
3. Sketch the shape of the opening you desire and then click Finish Sketch. The opening is then cut out of the roof extrusion.

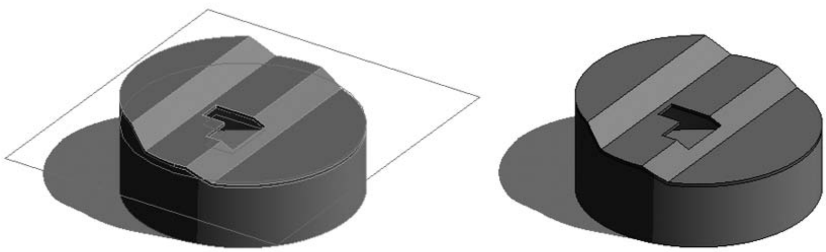
FIGURE 5.23
Sketching an area to void out for a roof opening



Another task you will most likely have to perform when creating the roof resembles using a cookie cutter on dough. Since the extrusion is rectangular or square, you may need to cut out a pattern from the rectangle to match your roof shape. You need to shape the edge of the rectangular extruded element to suit the shape of your structure. In this case, we are adding a void shape to cut the roof into a circular shape.

To cut a circular shape from the rectangular extrusion (see Figure 5.24), draw an inner circular boundary and then a boundary outside the overall extrusion. Then highlight the walls and, on the Options bar, click Attach. Once you select the roof element, the walls will automatically be attached to the underside surfaces of the roof.

FIGURE 5.24
Roof by extrusion with a circular pattern cut and the walls attached to the underside of the roof



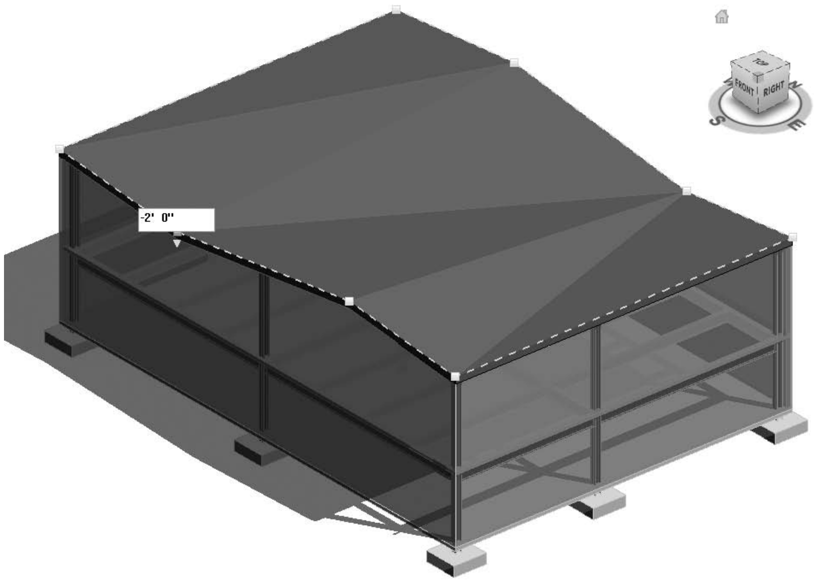
Warped Roof Deck Creation

Next we'll look at another type of roof system, flat roofs that slope from ridges to drains, which are found in the majority of typical commercial buildings. If you can learn to make these roofs, you will be able to model most of the new projects that come your way.

For commercial construction, many buildings will have a "flat" roof. In reality the roof is not flat but must slope slightly, usually at about ¼″ per foot from ridges to drains. This can be a big challenge to model and edit as you are working through the design process. Not only the roof diaphragm but the support framing as well will present complications, since all framing members are at different slopes with different end elevations. Until now the roof diaphragms and framing members were in one plane. As you will see in Chapter 7, the framing members under the warped roof decks can still be generated as a straight line.

As you can see in Figure 5.25, the roof low point, where the water drains, will be located at –2′-0″. The roof ridges are at 0′ in relation to the roof reference plane. Using the method of working in flat planes you could try to model a series of triangular surfaces to make the whole roof, but modeling would be cumbersome and time consuming. Revit Structure uses advanced roof creation tools, called subelements, to help you create this type of warped surface.

Figure 5.25
Roof creation subelements control the pitch of a sloping deck to its drains and ridges.



The general process looks like this:

1. Create the roof diaphragm.

2. Highlight the diaphragm, and the subelements become available on the Options bar. The four subelement buttons have the following properties:

   - The leftmost button (see Figure 5.26) puts you into an editing mode where you can pick the green lines and points and apply an elevation value.

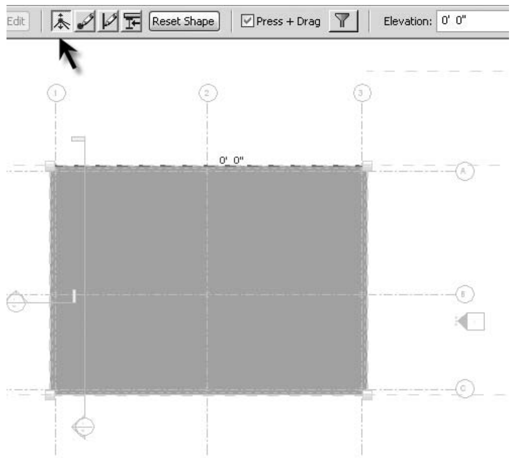


FIGURE 5.26
Highlight the deck and click the editing mode button.

   - The next button adds points anywhere on the diaphragm (see Figure 5.27). You can add an elevation value relative to the level to which you are attached. These points can then act as drain points or low points for the typical flat roof we are using as an example.

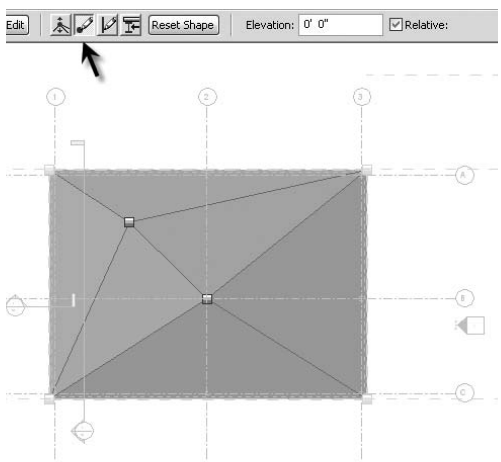
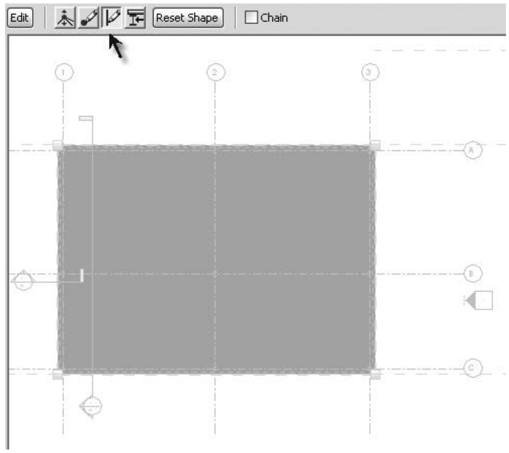


FIGURE 5.27
Adding elevation points to the roof diaphragm

- The third button adds lines to the diaphragm (see Figure 5.28) and then you can add an elevation value. These lines can act as ridge lines, or high points on your roof.

FIGURE 5.28
Elevation lines can act as ridge lines on the roof structure.



- The last button, on the right, allows you to add elevation lines to the diaphragm by selecting supporting beam members.

3. Use the Reset button to delete all of the added subelements and return the diaphragm to its original shape.

Real World Scenario

WHICH CAME FIRST, THE CHICKEN OR THE EGG?

In terms of roof creation, do you create your framing first and then apply a floor or roof diaphragm to it, or do you create the diaphragm first and then add the support framing? Truth is, there are so many conditions where either method is valid that you're wise to keep both in your tool chest.

In the case of a warped diaphragm, the answer is definitely the latter. In Chapter 7 you will see just why that is so.

Adding and Editing Roof Sub-Elements

Follow these steps to employ the sub-elements to form the warped roof deck:

1. Create a flat roof diaphragm. (For this example, use the Roof by Footprint option.)
2. Highlight what you have just created.
3. Click one of the sub-element buttons, then click on the roof diaphragm to add it.

4. When you have added the points or lines, click on the leftmost button to add or modify the elevation value for it.
5. To edit the elevations of the sub-elements, do one of the following:
   - Highlight the roof element, click on a line or point, click into the elevation value field, and change it as required.
   - Dynamically alter the shape by clicking and dragging the point or line. You can do this in a 2D or a 3D view.

Using Solid Blends and Sweeps for Deck Creation

Two other ways to create more exotic freeform types of roof shapes require that you use solid modeling techniques. The two solid modeling options you will consider here are for a sweep and a blended solid. You access these commands on the Modeling menu by clicking Create…

Using a solid sweep can also be useful for creating roof elements such as canopies that follow a varied path. Notice the profile lines and the path line that the profile follows in Figure 5.29. You must understand these two components to create this type of form.

A warped type of roof diaphragm often creates a smooth slope along its path (see Figure 5.30) from ridge to drain. In cases like that, the flat plane techniques that you learned earlier in this chapter will not work. An alternative is to use a solid sweep or a solid blend. Once you become accustomed to using the solid modeling features, you will be able to create nearly any shape imaginable (see Figure 5.31). You will undoubtedly face many challenging projects with many exotic shapes. We will cover these advanced solid modeling techniques in Chapter 19.

Figure 5.29
Barreled roof canopy created with a solid sweep

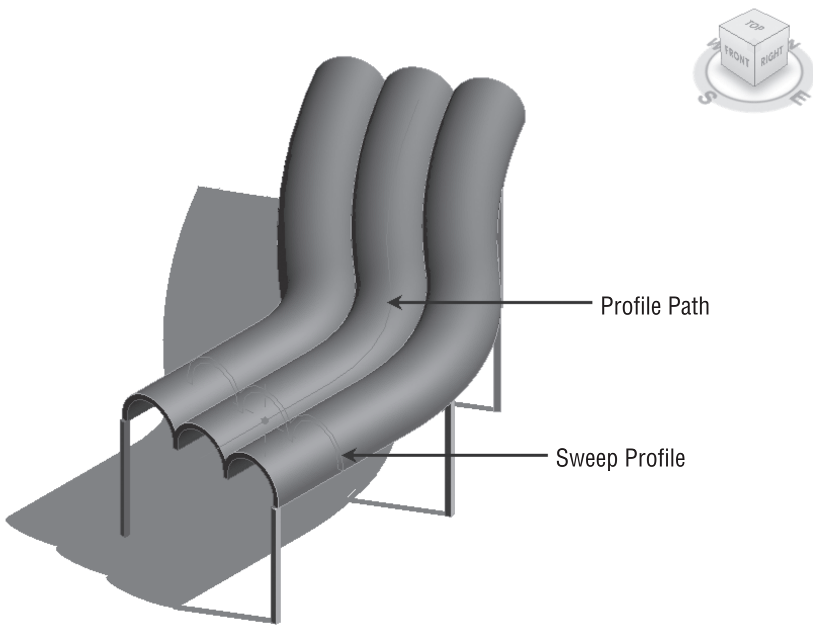


FIGURE 5.30
Wave-style roof diaphragm created by using a solid blend

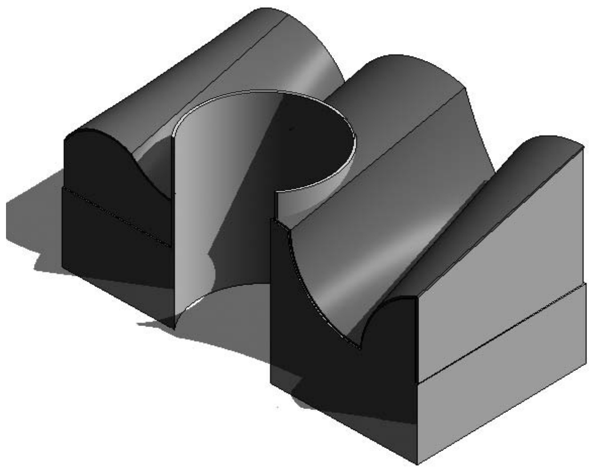


FIGURE 5.31
Freeform roof shape created using a solid blend

CREATING A ROOF DIAPHRAGM USING A BLENDED SOLID

Another interesting approach for creating a roof diaphragm is to use a blended solid form. The use of two profiles that "blend" together can make difficult shapes easy to create. In combination with a solid void to cut and shape the blend, you can create most shapes using these tools.

The basic approach is as follows:

1. In a plan view, create two reference planes, one at each end of the blended solid. Give them the names **1** and **2**.
2. Cut a section in front of the first reference line and stretch its extents past the second reference line.
3. Go to the section view you just created.
4. Click Create on the Modeling menu.
5. Choose Roofs as the category, and name the new element.
6. In the Family Editor, click Solid and then Solid Blend.

7. Set the first reference plane as the current plane.
8. Draw the profile of one side of the solid as a closed-line sequence with the thickness of the roof.
9. Click Edit Top.
10. Click Plane on the toolbar and select the second reference plane to make it current.
11. Draw the other second profile in the same way.
12. On the Design bar Click Finish Sketch, then Finish Family.

Attaching Walls and Columns to Floors and Roofs

An important feature of floors and roofs is that you can attach walls and columns to their underside. This is helpful in many cases, especially for roof conditions. A warped roof, for instance, with walls below would require editing the wall in elevation to match the roof warping. If the roof outline then changes during the design process, you would have to continually go back and fix the walls. That's the kind of busywork that you do not want to be spending your project fee fixing. You want to spend your fee concentrating on your design.

Column attachment to the floor or roof is also important. As you can imagine, with a warped or sloping roof your columns will each have a different elevation at their tops. Calculating each of those values would be time consuming, and the editing process would be grueling. Another alternative is to go into a section view, measure the distance to the underside of deck, and adjust it in the Column Element Properties dialog box. That too is time consuming and fairly inaccurate.

The procedure to attach walls and columns to your roof deck is straightforward:

1. In a 2D or 3D view, highlight the columns or wall.
2. On the Options bar, select Attach (or Detach if desired) and click on the floor or roof element to which you wish to attach the wall.

The walls will stay attached even if the form of the roof changes, as long as the roof remains located over the wall.

Real World Scenario

Flexing the Model Can Cause Unintended Results

Set a goal for yourself while creating your model: the elements in the model should be adequately constrained and have the ability to flex constantly. It can be intimidating when elements start moving around automatically due to the constraints to which they are attached. You have to keep an eye on those changes, and anticipate them by understanding the relationship between elements in Revit Structure. Keep a check on the various views that you have placed on your sheets, and make it a habit to quickly do a survey either on screen or of the hard copy before you send out your drawings to make sure elements have not moved in unexpected ways.

Openings and Depressions

An important consideration for any slab or deck is the ability to create openings and depressions. Slab openings are often shafts, for elevators, for mechanical ductwork, for light wells, and things of that nature. Depressions are often necessary for tile setting in bathroom or kitchen areas. If you want to keep the model accurate, you must address these elements.

To create the openings, you can edit the deck in the following way:

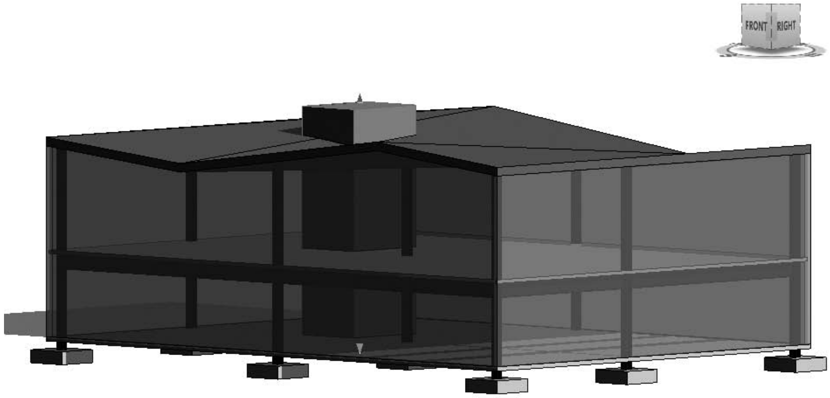
1. Highlight the slab, then click the Edit button on the Options bar.
2. Draw an enclosed polygon with the Sketch tools within the boundary of the slab where you want the opening.
3. Finish the sketch.

This is the simplest method. You can also make the opening flex with the beams that support the deck around it by locking them together. Put your plan view into Medium mode temporarily and lock the opening lines to the edges of the beam flanges (see Figure 5.32). That way, if the beam sizes change, the opening will change with them.

When adding core shaft and elevator openings, use the Shaft Opening tool to make your openings. The advantage is that the openings are automatically created in all floors and roofs that the shaft penetrates, and the openings are consistently in the same location floor to floor (see Figure 5.33).

Figure 5.32
Locking an opening to adjacent framing while in Medium display mode

FIGURE 5.33
Shaft opening penetrating floors and roofs of the structure



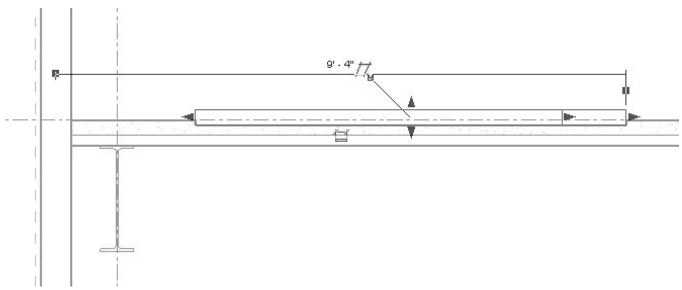
CREATE A SHAFT OPENING

Adding a shaft opening to your project is not too difficult, and works like this:

1. On the Modeling menu, click Opening ➢ Shaft Opening.
2. In the resulting Sketch mode, use the drawing tools to create the opening shape.
3. Click the Properties button to access the Element Properties dialog box.
4. Adjust the top and bottom constraints to set the vertical extents of the shaft.
5. Provide level offsets a bit below and above the roof.
6. Finish the sketch, and Revit Structure creates the shaft.

Another feature common to slabs that you will likely encounter is the isolated slab depression. For example, you may have thin-set tile less than an inch deep in the bathroom areas of a building (see Figure 5.34). A deeper set tile of 3″ to 4″ is also common. How you would model these depressions depends on the type.

FIGURE 5.34
Slab depression with the void highlighted in section



By creating and attaching a void extrusion to the slab or deck, you can hollow out the basic slab element. That will work fine for small depressions. Deeper depressions may require a lowering of the slab in that area. For that type of construction, you will need to make a new slab element and move it downward.

Creating Slab Depressions in Your Slab or Deck

To create a depression in a slab:

1. Go to a plan view of the slab.
2. On the Modeling menu, click Create.
3. Select Floors for the category and name the new family something descriptive like `Depression First Floor`. That puts you into Sketch mode.
4. On the Modeling menu, click Void Form ➤ Void Extrusion.
5. Click Lines and use the drawing tools to make the boundary of the depression.
6. Click Extrusion Properties on the Sketch menu.
7. Set the extrusion start and extrusion end values at 1″ (see Figure 5.35), so that the void extrusion goes a bit above the slab.
8. Set the extrusion end value equal to the depth of the depression.
9. Click Finish Sketch.
10. On the toolbar, click the Cut Geometry button.
11. Select the slab or deck and then select the void.
12. Click Finish Sketch, and Revit Structure creates the depression.

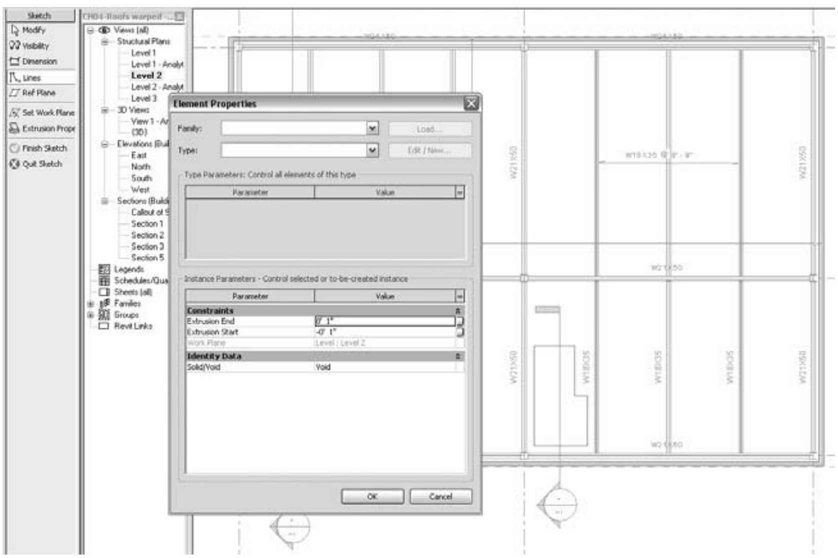


Figure 5.35
In the Element Properties dialog box, you set the vertical range of the void extrusion by adding an extrusion value in each direction relative to the floor level.

Exercise: Creating a Simple Sloped Roof with Dormers

In this exercise, you will create a roof diaphragm:

1. Open a new project.
2. Click Level 2 in the Project Browser.
3. On the Architectural bar, select Roof and then Roof by Footprint.
4. Choose Lines from the Sketch bar.
5. On the Options bar, click the Rectangle tool, then draw a rectangular outline.
6. Click on the reference plane and draw three vertical planes inside the rectangle.
7. Click Dimensions and create a dimension string connecting all five vertical lines. Highlight the dimensions and click the EQ button.
8. Erase the top and bottom lines.
9. Click Lines and draw four equal lines on the top and bottom, using the reference planes as a breakpoint guide.
10. Click Slope Arrow and draw the slope arrow over the two inner lines with the arrows facing the middle reference plane.
11. Highlight the slope arrows and click the Properties button on the Options bar. Set the Height Above level to 10´-0˝.
12. Click OK to exit, then repeat the process for the top line.
13. Highlight the remaining lines. On the Options bar, check the Defines Slope box.
14. One by one, highlight the other lines and click into the angle symbol. Set the slope to 30 degrees for the horizontal lines and 50 degrees for the vertical lines on the side.
15. Click Finish Sketch to finish the roof diaphragm.

The Bottom Line

Create a slab-on-grade with dropped slab edges. Using the Slab command, you saw how to create slabs-on-grade and to apply dropped edges to them with the Slab Edge function. You also learned how to edit the profile file for slab edges in order to create new types.

Master It You have a new project and have to add a slab-on-grade and slab edge at the bottom level. How do you do it?

Work with floor decks. You learned different methods for creating roof diaphragms and for editing them during the design process. You learned to create composite decks and how to create their geometry through sketching. You also learned to attach columns and walls to the roof diaphragm.

Master It Your project requires the metal deck to stop at the edge girder and the concrete to extend out one foot beyond to meet the inside face of the metal stud framing on the exterior. How would you create that?

Work with warped roof decks. You learned different methods for creating roof diaphragms that are not just planar. Methods include using the slab subelement tools as well as creating more exotic roof shapes with solid modeling tools.

Master It Your project has a main ridge line across the middle of the roof with two drains on two edges of the roof diaphragm ⅓ points of the edge distance. How would you create it?

Create openings and depressions in your floors and roofs. You learned how to create shaft and incidental openings, and how to add depressions to the slab.

Master It On a multistory building, you want to add shafts to the core areas for stairs and elevators. How would you do that?

Chapter 6

Walls

This chapter focuses on the creation of walls in Revit Structure. Walls are one of the most profound features of Revit. As a wall is modeled, all aspects of that wall's functions are considered—the structural usage, the type of wall, and even architectural considerations. Also within the initial design of a wall, the cover for reinforcing is specified. Learning how to create and work with wall systems is a must, but it is easy.

Walls, like slabs, floors, and roofs, are unique in the fact that they are considered system families. This means they are completely controlled, manipulated, and created within the current structural model. You do not insert a wall as a separate entity into a model. It also means that walls can change dynamically as the model changes. When a wall is "constrained" between two floors, for example, the actual dimensional difference between the floors influences the height of the wall. If a floor height changes for any reason, so does the wall height.

It is important to note at the beginning of the section that wall types that are to be repeated through many projects should be added to the overall company template. To learn more about template creation, refer to Chapter 17. If a wall has been created in another project, the wall style can be carried into the current model by simply copying and pasting the wall into the current model from the existing one. Although this is often frowned upon in drafting applications, Revit Structure is a database that fully allows copying and pasting without residual, unexpected side effects such as extra layers, blocks, and even shape files, which can quickly corrupt a drawing.

The first type of wall system to be covered will be a basic wall. Don't let the name fool you—it can be anything from a plain concrete wall to a complex wall system with structural and architectural elements combined. The basic wall type, however, will certainly be the most common wall type used. Revit Structure comes "out of the box" with 14 predefined types. Some of these wall types will be fine to use as is, and some of them will be used as a starting point for a more complex wall system.

In this chapter you will learn to:

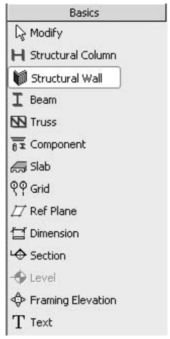
- Place walls in your model
- Create new walls
- Modify walls in-place

Placing a Wall in Your Model

To add a basic wall, you first need to be in either a plan view or a 3D view. You should also have some levels other than Level 1 and Level 2 defined, but it is not imperative. After all, the power of Revit Structure lies in the fact that changes are affected globally and are propagated throughout

the model. Also, it is recommended that you place walls in a plan view and not a 3D. You will have much more control, and you are less likely to make mistakes. To the left of the Revit Structure interface is the Design bar. Walls are normally created using this interface. On the Basics tab of the Design bar, you will see the Structural Wall icon (see Figure 6.1).

Figure 6.1
The Basics tab of the Design bar

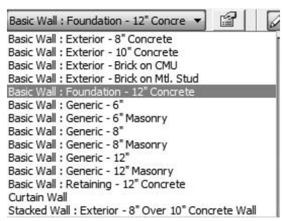


The command process in Revit Structure is quite intuitive. You select your command (in this case the Structural Wall command) and Revit Structure will populate the Options bar (see Figure 6.2). When you exit the command, the Options bar for that specific command goes away. This greatly reduces the clutter found in most traditional CAD programs, where every toolbar and icon is placed on the screen at once.

Figure 6.2
The Options bar

On the Options bar, several features will stand out. The first of the features is what is called the Type Selector (see Figure 6.3). This is a list of the defined wall systems found in this specific project. This list can simply be just a few wall types, or it can be extremely long depending on the size and stage of your project. We recommend that this list be as short as possible, and that you adhere to company-wide naming conventions. There is nothing worse than having duplicate wall types to choose from. As you start the Wall command and the Options bar populates, the first thing to do is select a wall type from the Type Selector.

Figure 6.3
The Type Selector allows you to choose the wall you need to add to the model.



Once you select the item from the Type Selector, Revit Structure continues to provide choices from the Options bar. The first set of controls determines how you will place the wall in your model. You can either match existing objects in the model, or you can click the Draw icon and draw the wall from scratch.

The Draw Option

By clicking the Draw icon, you are telling Revit Structure you want point-by-point placement of your wall. You will be prompted to pick a point in the view window: Click to Enter Start Point. Because Revit Structure does not have an actual command prompt, it has a status bar to guide the user. This comes in handy for more convoluted commands. Once the first point is selected, you will see a blue alignment line. This takes the place of the traditional “ortho,” or polar tracking. If your cursor is straight in any direction, your wall will snap to 0, 90, 180 or 270 degrees. You can configure additional snap settings (see Figure 6.4).

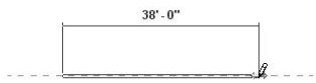


FIGURE 6.4
Alignment lines are basically the replacement for the “ortho” function in AutoCAD.

As you sketch the wall, a blue dimension appears. This is called a *temporary dimension* (see Figure 6.5). Instead of having to eyeball a second point, you can type in a distance instead. Since Revit Structure is factored automatically by architectural units, you can type in the value without the foot mark. For example, if you wanted a wall that was 50´-0˝ long, you could simply type in **50**. Revit Structure accepts this value is in feet. If you wanted the wall to be 50˝ long, you would have to type in the inch mark. Also, Revit Structure will accept equations as you draw the wall. If you wanted a 50´-0˝ long wall, for example, you could type **=25*2**. Once the second point is selected, Revit Structure will place the end at the result of the equation. Normally, the process will be to sketch the walls, but if geometry is already placed in the model, it may be tempting to just select that geometry.

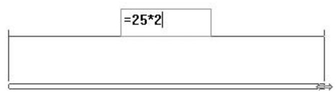


FIGURE 6.5
You can type temporary dimensions, or enter a mathematical formula by beginning the formula with an equals sign.

The Pick Option

By clicking the Pick icon, you are telling Revit Structure that your method of wall placement will be to select existing items in the model. You will typically use this approach when you are referencing an architectural model in the structural model. In most cases, this will be a 2D architectural CAD file. We recommend that you proceed with caution while selecting lines using this method. Often the CAD data will contain walls that are not 100% straight. If this is the case, and you are adding a new Revit Structure wall by picking the CAD lines, Revit Structure will report the issue (see Figure 6.6).

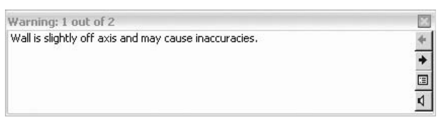
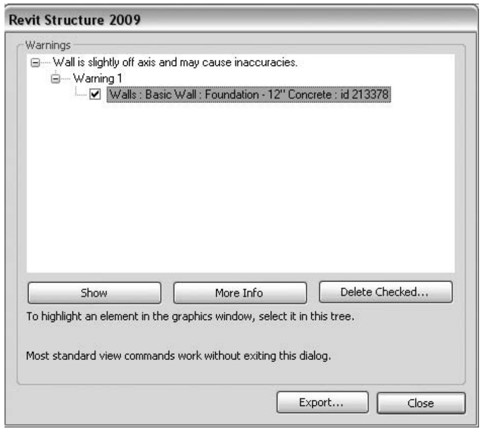


FIGURE 6.6
The “off axis” warning informs you something is amiss.

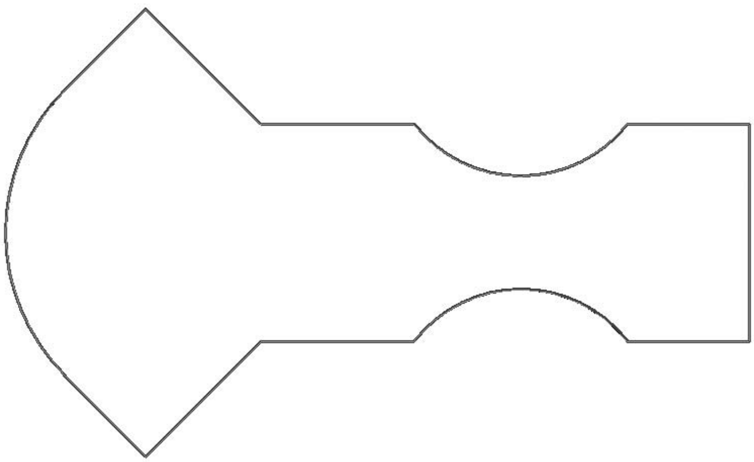
Revit Structure will generate a warning that can be ignored. Although it is tempting to ignore, we suggest that you ensure your wall is being drawn straight and accurately. This warning can also be expanded to see exactly which wall generated the error. Revit Structure will list the error and then break the list down to the specific item(s) that are suspect. Once the items are selected, you will be able to see the "crooked" walls in the model. At that point, you can delete the offending item. Again, we recommend that you go ahead and delete the wall instead of trying to adjust it or leaving it in the model crooked. See Figure 6.7 for an illustration.

FIGURE 6.7
You can click the expanded warning button to see a full preview of the problem.



The Pick icon is also useful for adding walls to a sketch. Many times it is useful to place drafting lines in place of walls to establish important layout dimensional constraints. In preliminary stages of a project, it is sometimes beneficial to approach a model in this mind-set. Sketching the lines and then adding the walls afterward in many cases is much easier and actually safer (see Figure 6.8). For more information on drafting lines and sketching, see Chapter 9.

FIGURE 6.8
Drafting lines



Later, once the perimeter dimensions have been established, you can place the walls by selecting these lines. Simply start the Structural Wall command, then click a line. A useful method is to use the Tab key to pick lines. While running the Wall command, you can simply hover your cursor over a single line or element. Once the line highlights, press Tab. All of the lines or elements that are similar and that are connected will become highlighted (see Figure 6.8, earlier).

Since you are basically drawing in 2D, what about the height of the wall? Don't you need to "extrude" it? The answer is no. You can set the height based on either a level or an increment. The results should look like Figure 6.9.

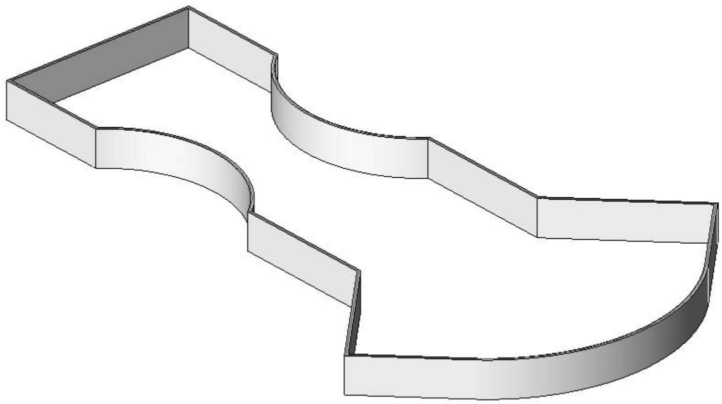


FIGURE 6.9
The 3D result of picking lines. Although this is a nice view, always be sure your Loc Line constraint is set to the justification you need.

Real World Scenario

USING LINES AS A CRUTCH

You may find as you start modeling your footprint, if it is an odd shape, that the best way to be sure your geometry is where you want it is to simply start using drafting lines as a guide. Once the drafting lines are in place, you can then start selecting those lines to add your walls. In many cases this has proven to be a good policy, especially in the beginning stages of your relationship with Revit Structure.

Top/Bottom and Height/Depth Constraints

As mentioned earlier, walls are special based on the fact that they can be constrained to the project's levels. This means that the base and the top can both be "locked" to an elevation (see Figure 6.10). In other words, you can set it and forget it.

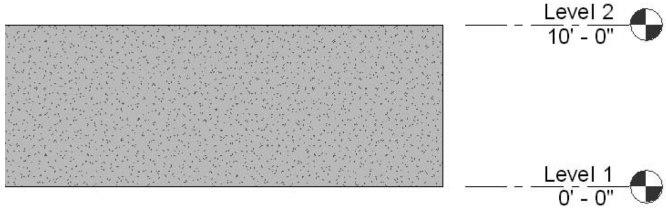


FIGURE 6.10
On the Options bar, you will see the top and bottom constraints.

To do this, glance at the Options bar. It has a drop-down to handle the Height and Depth constraints (Figure 6.11). It is automatically assumed, unless specified otherwise, that the depth will be at the current level that you are drawing the wall. The height is normally unconstrained as a default that can be changed on the Options bar. Once you select a depth or height, you can choose the level it will be constrained to in the adjacent drop-down. If the height or the depth is unconnected, you can simply type a distance. We recommend that you specify a level in lieu of an unconnected height when possible. Of course, once you start sketching the wall or selecting geometry to use as a guideline, you need to set a justification on which the wall will be located.

Figure 6.11
On the Options bar, you will see the Height and Depth settings.

Location Line (Loc Line)

Possibly the most important item on the Options bar is the location of the wall (see Figure 6.12). This can ripple through a project in a negative or a positive way. When you are laying out a wall, be aware of the fact that, while the justification of a wall can always be modified, it is not always easy to do so toward the end of a project.

There are six choices for locating the justification of the wall, as explained in the following sections.

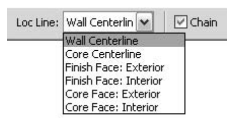


Figure 6.12
On the Options bar, you will see the Loc Line drop-down.

Wall Centerline

The wall centerline calculates the centerline of the wall based on the two outermost faces (see Figure 6.13). It could be gypsum or some lightweight finish, or it could be a structural element such as CMU or concrete. While this is fine for a single element type wall such as a concrete foundation wall or concrete masonry unit (CMU) wall, it is not recommended for a compound wall such as a CMU wall with a brick façade. Also, walls with finishes don't lend themselves well to this justification.

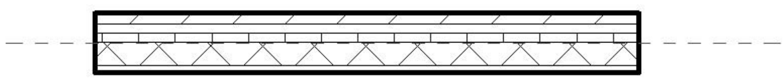
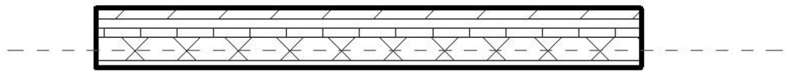


Figure 6.13
On the Options bar, you will see the wall centerline.

Core Centerline

The core centerline justification allows you to justify the wall about the centerline of the structural element in the wall (see Figure 6.14). In the example used here, the compound wall is 8″ CMU with 2″ of rigid insulation, a 2″ air space, and a 4″ brick façade. The justification of the wall is based on the center of the CMU. This helps in the placement of the structural column grid.

FIGURE 6.14
Here, the alignment line is centered on the wall's core centerline.



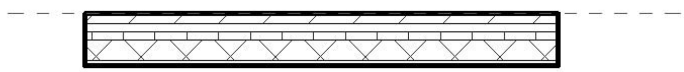
FINISH FACE EXTERIOR

This one is self-explanatory. The justification of the wall will be to whichever face is to the outside. As you are placing the wall, however, the inside and outside may not be what you expect. You have the option to press the spacebar to change the interior and exterior of the wall. To do this, use the following steps:

1. Select the Structural Wall command on the Basics tab of the Design bar.
2. Select a compound wall type such as Basic Wall: Exterior Brick on CMU.
3. Change the Loc Line to Finish Face Exterior.
4. Click the first point for the placement of the wall.
5. Press the spacebar.

Notice the wall will flip about the justification axis, as illustrated in Figure 6.15.

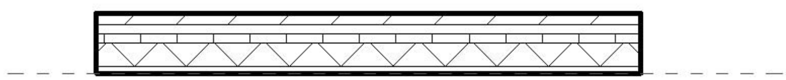
FIGURE 6.15
This wall is justified about the finish face exterior.



FINISH FACE INTERIOR

This justification allows the wall to be placed using the finished face of the outside of the wall. This is normally an architectural finish and is probably the least common of the six choices, but it is still sometimes necessary (see Figure 6.16).

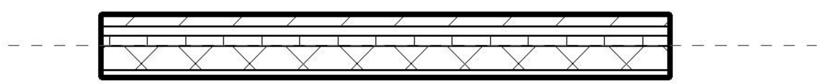
FIGURE 6.16
This wall is justified about the finish face interior.



CORE FACE: EXTERIOR

This option justifies the wall to the face of the structural element of the wall. The face it chooses is the side facing the exterior. Usually this is in the middle of the wall core, as shown in Figure 6.17. This allows you to justify the wall as if there were no architectural elements, and the insulation, air space, and brick are ignored.

FIGURE 6.17
This wall is justified about the core face's exterior.



Core Face: Interior

Like core face exterior, this option will justify the wall based on the inside face of the structural element. Any finishes on the inside of the wall will be ignored. See Figure 6.18.

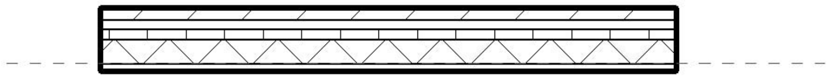


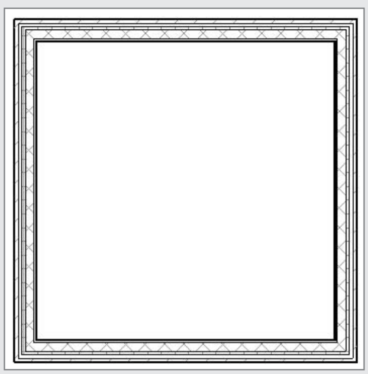
Figure 6.18
This wall is justified about the core face's interior.

Exercise: Justifying Walls

As a practice for the wall justification, perform the following steps:

1. Go to a Level 1 floor plan view.
2. On the Basics tab of the Design bar, select the Structural Wall command.
3. Select the wall Basic Wall: Exterior – Brick on CMU.
4. Change the Height to Level 2.
5. Select a start point.
6. Change the justification to Finish Face Exterior.
7. Check the Chain button.
8. Draw a wall 25´-0˝ straight to the right. If the justification is incorrect, press the spacebar.
9. Draw another wall 25´-0˝ straight up. If the justification is incorrect, press the spacebar.

Although this is simply a box, it is important to understand how Revit Structure works, and more importantly what it expects from the end user. Yes, placing walls is simple, but an eyes-open approach is needed to get the desired results. Although all of these options can be fixed later, getting the most accurate placement initially is key to a successful project. See the following illustration for the finished walls.



If it is your choice to sketch the walls, there will be some additional tools available on the Options bar. These options will allow you to draw the walls in the geometry you need.

Sketch Tools

It does seem redundant to keep drawing walls piece by piece. After all, every drafting program available allows users to draw any shape they choose. Revit Structure is no different. With the proper height and base constraints and the proper justification, Revit Structure will allow you to "sketch" any desired shape. As a default, the Line choice is current for the placement of any new wall. To the right of the Line choice is the Rectangle choice.

The placement of a wall using the Rectangle choice involves these steps:

1. On the Basics tab of the Design bar, click the Structural Wall button.
2. Set the height and Loc Line to the desired settings.
3. Click the Rectangle button.
4. Select the first point for the four walls.
5. Move your cursor.

You will see the walls form. They may be positioned incorrectly. At this point you can press the Tab key. This will flip the orientation of the wall. Unfortunately, you have to eyeball the dimensions.

The placement of a wall using the arc passing through three points choice involves these steps (see Figure 6.19):

1. On the Basics tab of the Design bar, click the Structural Wall button.
2. Click the arc passing through three points button. (This command is similar to the Arc Start-End-Direction command in AutoCAD.)
3. Click the first point of the arc.
4. Click a second point of the arc.
5. Either specify a third point or type in a radius. Remember that if the justification is incorrect, you can press the Tab key to flip the wall.

There are other sketch options as well (see Figure 6.20).

FIGURE 6.19
Arc passing through three points

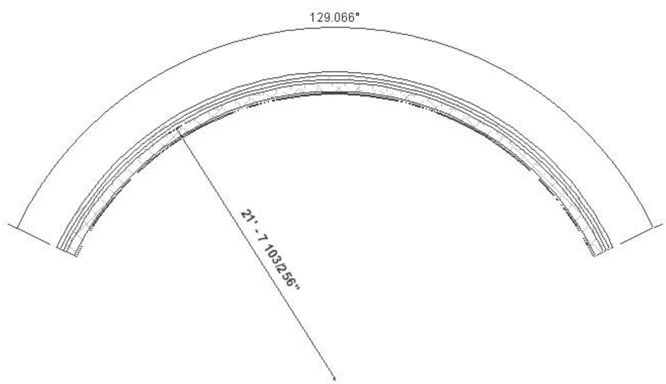
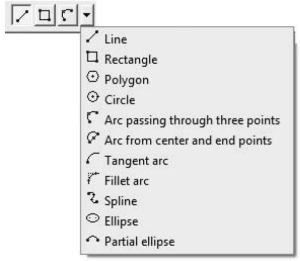


FIGURE 6.20
Sketch options

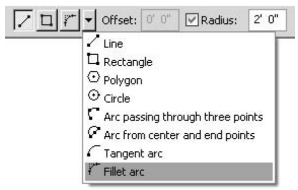


Let's try the last choice. Revit Structure does not have an actual fillet command that will allow you to select the wall and add a radius at a later time. It does, however, have a Fillet Arc command built into the Type Selector drop-down during placement.

The placement of a wall using the Fillet Arc option is as follows (see Figure 6.21):

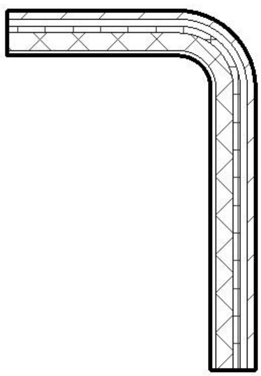
1. On the Basics tab of the Design bar, click the Structural Wall button.
2. Place two L-shaped walls.
3. Restart the Structural Wall command.
4. Select the Fillet Arc option from the drop-down.
5. Select the Radius option and enter a distance of **2′-0″**.

FIGURE 6.21
The Fillet Arc setting on the Options bar



6. Click the two walls, and the fillet arc is added (see Figure 6.22).

FIGURE 6.22
The fillet arc once it has been added to the walls

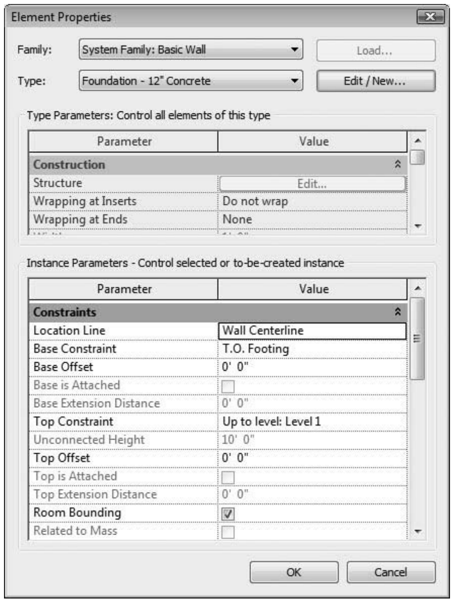


Now that some walls are drawn in the model, it is time to see how we can further manipulate them. Simply placing a wall and then changing the visible options is literally just scratching the surface of what can be done. There is an entire dialog box filled with options and settings we can use to configure our walls. Once the wall(s) have been added to the model, you can modify the element properties of the wall itself.

Element Properties

By placing a wall and then selecting the element properties, you gain access to every setting that was available in the Options bar (see Figure 6.23). These options are called *instance parameters*. This means that the options will only pertain to the wall that is selected. If you would like to change every wall of that type in the entire model, you can access the Type Parameters by clicking Edit/New. This is common not only in walls, floors, slabs, and roofs but in most families in a Revit Structure model (see Figure 6.23).

FIGURE 6.23
In the Element Properties dialog box, the first set of parameters you see are the instance parameters.



INSTANCE PARAMETERS

To gain access to the instance parameters of a wall using the Element Properties dialog box, follow these steps (see Figure 6.24):

1. Select a wall or a group of walls.
2. Click the Element Properties button on the Options bar.
3. In the resulting dialog box, you will see all of the instance parameters.

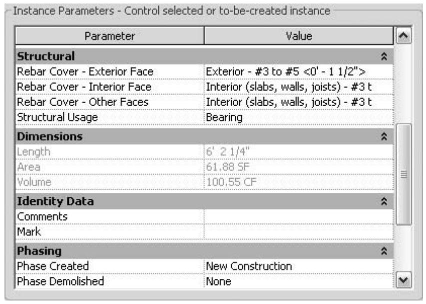


FIGURE 6.24
Instance parameters change only the parameters of the selected wall.

> ### PARAMETER CATEGORIES
>
> The parameters for the walls are broken down into categories. One category is the Structural category. This will determine the cover of the reinforcing as it is placed into the model as well as the structural usage. This item, along with the Phasing and the Analytical model, will be discussed in Chapter 15.

TYPE PARAMETERS

Type parameters of a wall are settings that define what a wall is. These parameters, if modified, will influence every wall in the model. So, if you have a CMU wall and you change the type parameter called Material to Brick from Concrete Masonry Units, you have just forced every wall of that type to be a brick wall. So, given that, you must make changes deliberately. To access these properties, first select a wall, then click the Element Properties button on the Options bar. Next, click the Edit/New button at the top of the dialog box, as shown in Figure 6.25.

Once you click Edit/New, the Type Parameters dialog box opens (see Figure 6.26).

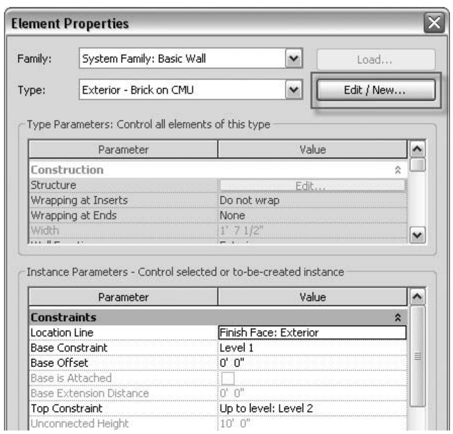


FIGURE 6.25
The Edit/New button allows you to access the type parameters.

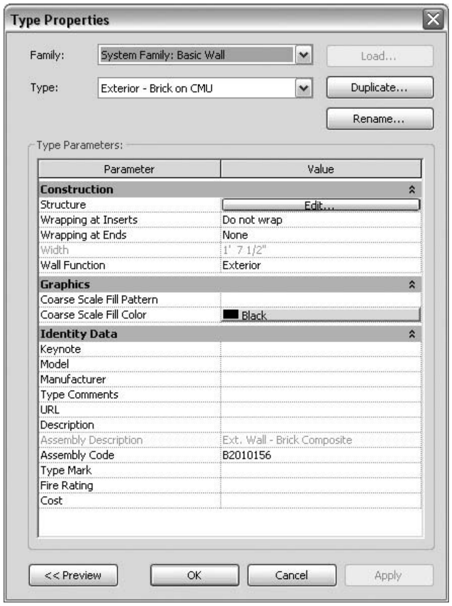


FIGURE 6.26
The Type Parameters dialog box allows you to edit every wall for that type in the entire model.

Follow these steps to edit type parameters in a wall system:

1. Select a wall to be modified.
2. Open the Element Properties dialog box.
3. Click Edit/New.
4. Click Duplicate (see Figure 6.27).
5. Rename the wall system to something that makes sense.

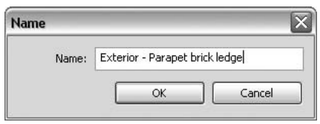


FIGURE 6.27
The name dialog box appears once you click the Duplicate button.

It is important to get into the practice of creating new wall systems in this manner. Also use type parameters when you plan to either change the current wall system globally or create an entirely new wall style. This sequence steps you through the process:

1. Click the Preview button at the bottom of the Element Properties dialog box. The dialog box expands to display a graphic of the wall (see Figure 6.28).

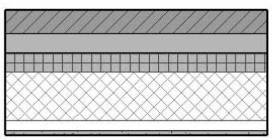


FIGURE 6.28
The preview allows you to see the wall.

2. Beneath the preview is a View drop-down. Select Section: Modify Type (see Figure 6.29).

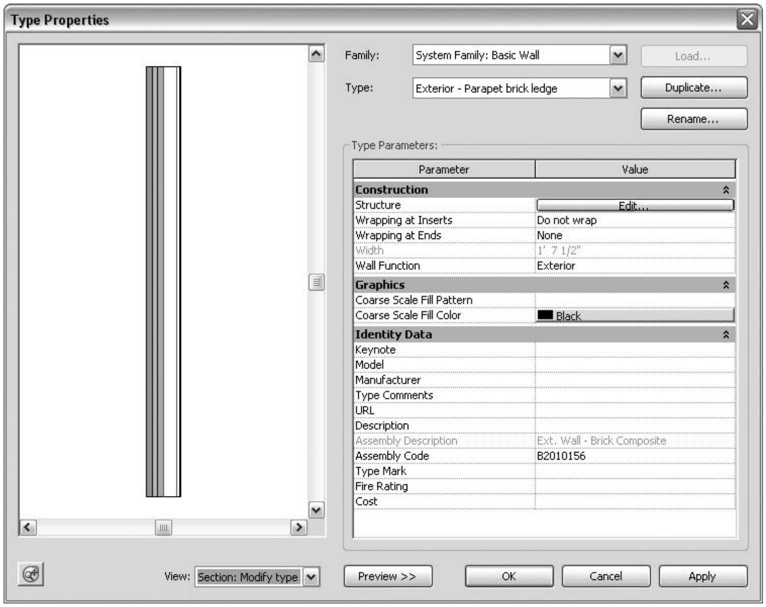


FIGURE 6.29
Modify Type Attributes allows you to really dig into the wall.

Another way of accessing the type parameters of a wall type in Revit Structure is to find the wall types in the Project Browser (see Figure 6.30). As you scroll down the list, you will come to a portion in the browser where the families are listed. If you drill in deeper to the families, you will see the Walls category. Here, the walls are broken out into the three categories: Basic, Curtain, and Stacked. If you drill in even further, you can simply double-click on a wall to access the type properties. This method is slightly different than selecting a wall from the Design bar—with this method you only have access to the properties that will change all of the walls of this type in the entire model.

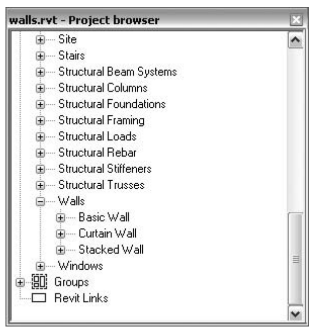


FIGURE 6.30
Finding a wall in the Project Browser

To create a new wall type from the Project Browser, perform these steps:

1. Scroll down the Project Browser until you come to the Families category.
2. Find Walls.
3. Select Basic Wall and expand the list.
4. Open the Basic Wall category and find the wall you are looking for.
5. Right-click and select Duplicate.
6. Rename the new wall style.
7. Double-click on the new wall style.

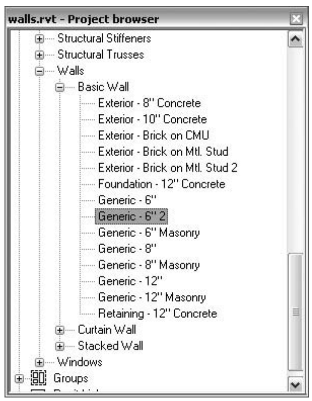
Notice you only have access to the type parameters. This is a good way to ensure you are only editing the proper wall. As mentioned earlier, there are 14 walls to choose from in the out-of-the-box Revit Structure, but you will almost certainly need to start creating your own walls at some point. Usually "some point" is the first stage of your first project.

Creating a New Compound Wall

Now that you have the basics down, it is time to create a new wall type. Depending on how you plan to use the wall type, this can be either easy or somewhat difficult. Keep in mind that the amount of effort you put into the wall at this stage of the model can have positive effects down the line when it comes time to add sections and elevations.

Let's begin with a very simple wall. In the Project Browser, scroll down to the Walls category and drill into the Basic Wall category. Right-click on Generic 6″ and duplicate the wall (see Figure 6.31).

FIGURE 6.31
A new generic wall



Use the following procedure to make the new wall (see Figure 6.32):

1. Double-click on the new wall.
2. In the Type Parameters section, click the Edit button in the Structure row.

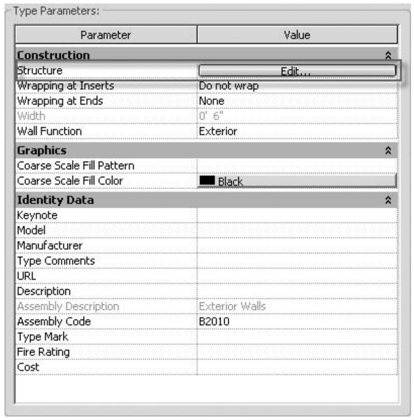


FIGURE 6.32 Edit the structure by clicking the Edit button in the Structure row.

3. In the Edit Assembly dialog box, make sure the Preview window is open and that the view is set to Section: Modify Type Attributes. This is important because otherwise you couldn't access some of the buttons we will be using (see Figure 6.33).

Note that before you start modifying this wall, you cannot escape or undo. You must plan each change carefully. Pressing the Esc key will cause you to lose all of your changes. If you make a mistake, simply click the Cancel button and redo the procedure. This does take practice and patience.

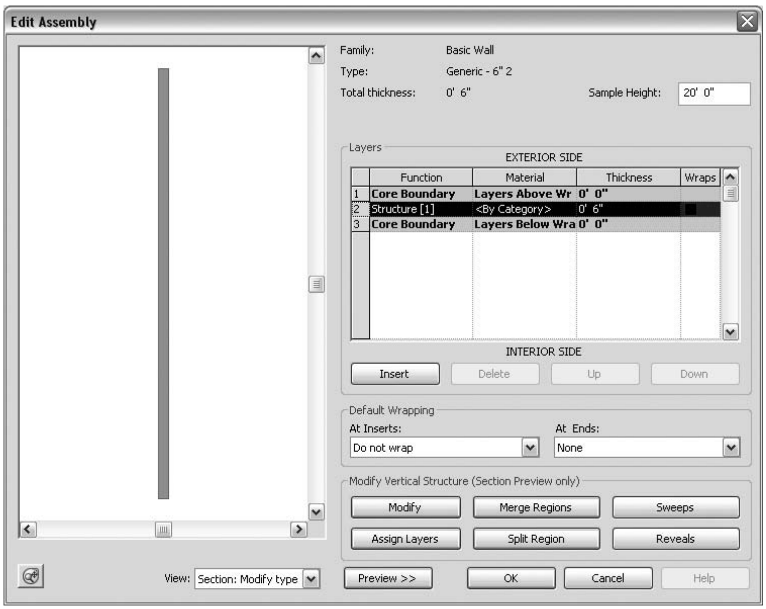


FIGURE 6.33 The "layers" that comprise a wall

You will see an area called Layers. There are two core boundaries that surround a structure. When you build a wall, anything that goes between the core boundaries is considered either *structure* or *substrate*. Anything that goes to the outside of these boundaries is considered *a finish or an air barrier*. Also, above the layers you see EXTERIOR SIDE and below them you see INTERIOR SIDE. This information becomes important when it is time to set the justification (see Figure 6.34). The following procedure will step you through the process:

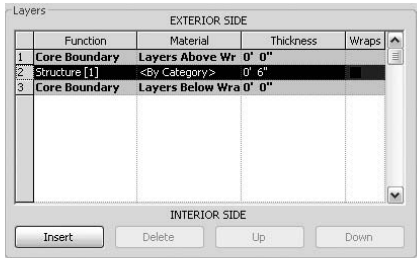


FIGURE 6.34
The Layers determine what materials will be applied to the wall, along with the thickness of each.

1. Click Structure in the Material column. An ellipsis button will appear in the right side of the column (see Figure 6.35).

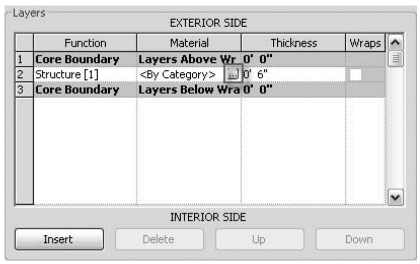


FIGURE 6.35
Materials Builder button

2. Click the ellipsis button to open the Materials dialog box (see Figure 6.36).
3. In the Materials section, select the material called: Masonry – Concrete Masonry Units.
4. Click OK.
5. In the Thickness column, assign a thickness of 8″.
6. Click on the number 1 row (core boundary).
7. Click the Insert button located below the Layers area.
8. Change the Structure function to Thermal / Air.
9. Click into the materials.
10. Select Misc. Air Layers – Air Space.
11. Click OK.
12. Give it a thickness of 2″. Notice the preview is starting to change, as shown in Figure 6.37.

FIGURE 6.36
The Materials dialog box

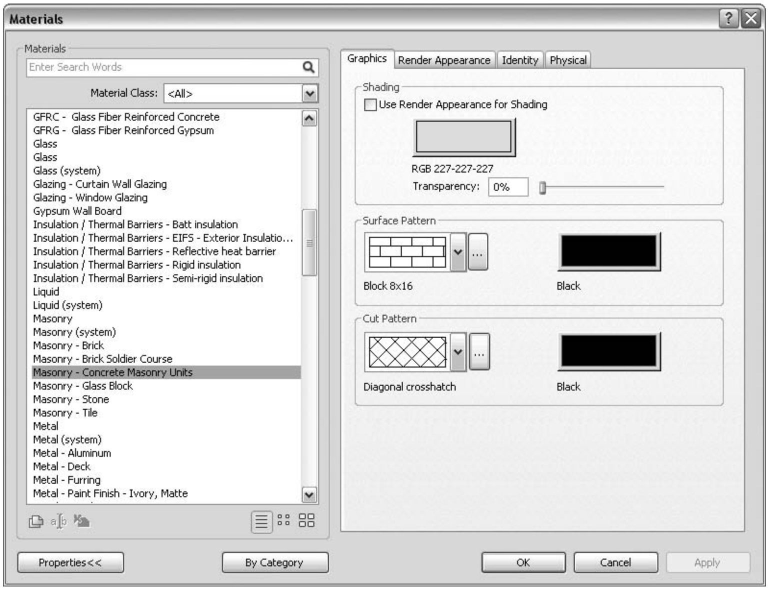
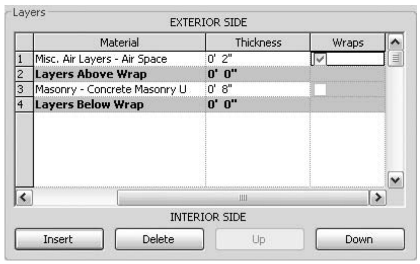


FIGURE 6.37
The added air space layer



13. Select the top row, and click the Insert button again.
14. Change the function to Finish 1.
15. Change the material to Masonry – Stone.
16. Change the Thickness to 4″ (see Figure 6.38).

In the preview, notice if you press your wheel button, you can pan around (see Figure 6.39). If you wheel in and out, you can zoom. Zoom in closer and to the bottom. We are going to split and merge the bottom 3′-0″ of the wall to create a compound wall situation.

FIGURE 6.38
Adding a stone veneer

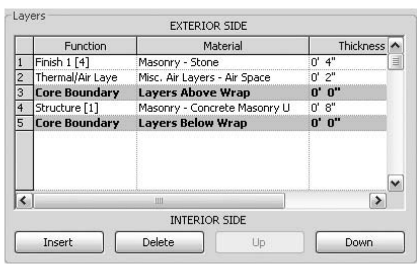


FIGURE 6.39
Using the preview window

Underneath the Layers area where you were just working, you will see the Modify Vertical Structure area (see Figure 6.40). This would not be available if the preview was not set for Section. Since it is, you can chop the wall up. Before you do, though, click OK to preserve your work up to this point. The following procedure will step you through the next process:

FIGURE 6.40
The Modify Vertical Structure button

1. Click the Edit button.
2. Click the Split Region button.
3. Your cursor turns into a knife. Revit Structure will add a temporary dimension to your cursor. Move 3´-0˝ up the wall, and split the 4˝ stone material (see Figure 6.41). This may take a couple of passes.

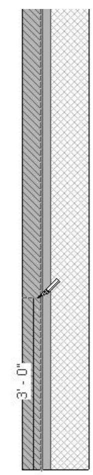


FIGURE 6.41
Split the material using the knife tool.

4. In the Layers section, add a new layer, specifying Finish 2 and Masonry – Brick (see Figure 6.42). Do not give it a thickness.

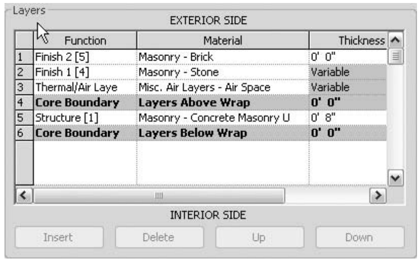


FIGURE 6.42
The wall once the masonry brick layer has been assigned

5. Using the Split Region command, split the air space at the same 3′-0″ height.

6. Click the Assign Layers button.

7. Make sure the brick material is selected in the Layers section.

8. Click the lower 3′-0″ of the stone (see Figure 6.43).

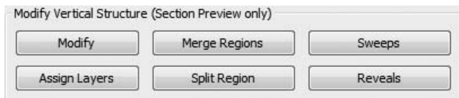


FIGURE 6.43
The Assign Layers button will allow you to add the new material to a section of the wall.

9. Click the Merge Regions button.

10. Hover your mouse over the line between the brick and the air space, below the 3′-0″ cut line. You will see that an arrow forms. The direction in which the arrow points is the direction in which the material will merge. Merge the brick into the air space (see Figure 6.44).

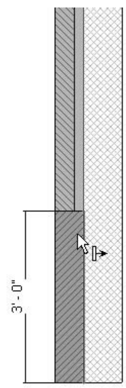


FIGURE 6.44
Merge Regions allows you to join two materials together.

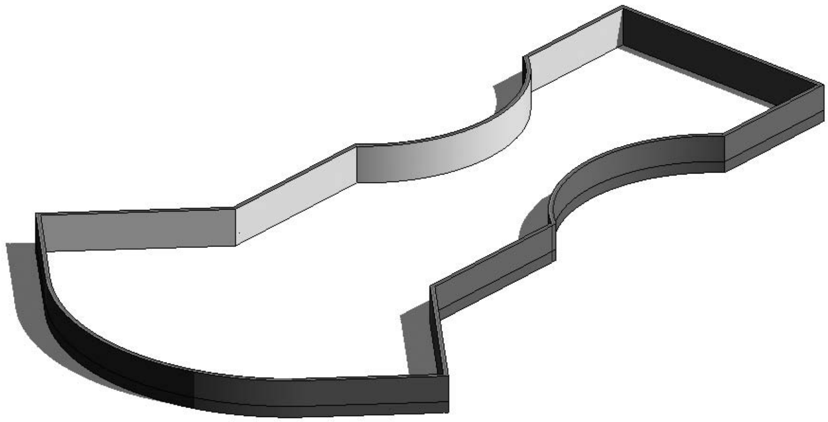
11. Click OK.

12. Click OK again.

You have a new wall type.

In some cases, the architectural façade extends above the core structure to form a small parapet. You can do this in Revit Structure. First, place some instances of your new wall in the model. It has been our experience that, just because it looks correct in the editor, it may not in the model. It is also wise to test how the wall is joining in corners (see Figure 6.45).

FIGURE 6.45
The new wall



In many cases, the architectural finish will extend above the bearing of the wall to form a small parapet for drainage blocking or to conceal structural elements. This can also be accomplished in Revit Structure with the following steps:

1. In the Project Browser, double-click on the new wall style, or open an existing compound wall.

2. Click Edit/New.

3. Click Edit in the Structure row.

4. Zoom in to the top of the wall.

5. Under the Modify Vertical Structure (Section Preview Only) area, click the Modify button (see Figure 6.46).

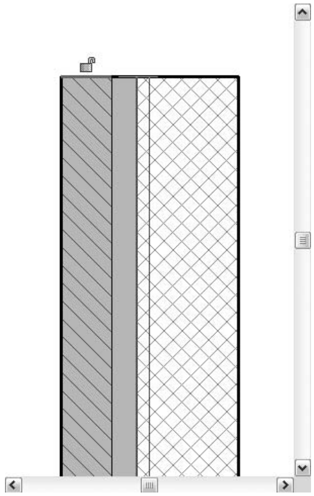
6. Select the top line on the brick or stone. Notice the padlock.

7. Unlock the padlock by clicking on it.

FIGURE 6.46
The Modify button allows you to access the vertical structures.

This releases the top wall constraint so that it's independent of the wall structure (see Figure 6.47). Once the wall is placed in plan, the wall can be edited in either a section or in a 3D view. Again, we recommend that you make modifications on a sectional view to maintain accuracy. It is also a good idea to set up a parapet level and extend this item to a level. You will have to use the Align and Lock feature to establish a dynamic link between the level and the top of the brick parapet. Once the wall is selected, you will see two blue arrows. One controls the façade and the other controls the structure. This indicates that the wall has been successfully released.

FIGURE 6.47
Unlocking the top

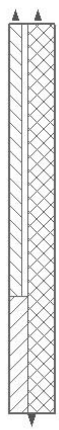


8. Click OK in this dialog box.
9. Click OK again.

This puts you back in the model. If you have this wall drawn, cut a section through it.

In the section, select the wall. See the two blue arrows? They can now be independently moved, as you can see in Figures 6.48 and 6.49.

FIGURE 6.48
The wall section.



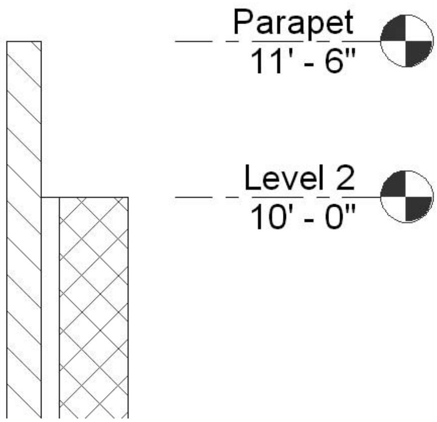


FIGURE 6.49
The parapet

Once the wall is physically placed in the model, the battle is only half over. Revit Structure has many functions that allow you to modify the wall once it is in place.

Modifying the Wall

Once a wall is placed in a model, additional functionality is often needed. Adding an opening and allowing the wall to attach itself to a sloping slab or a roof are a couple of tasks that need to be done. Also, creating openings and penetrations are sometimes a must.

PENETRATIONS

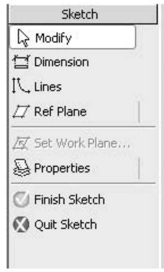
Penetrations are performed after a wall is placed in the model. This is usually done in an elevation, and can be accomplished in two ways. The first method is to physically edit the profile (see Figure 6.50). To add a penetration, as well as step the wall down, perform these steps:

1. Add a wall to the model. It can be a new wall or a wall that has been predefined.
2. Create an elevation looking at the wall.
3. Select the wall.
4. On the Options bar, you will notice some controls. Click the Edit Profile button.

FIGURE 6.50
Click the Edit Profile button on the Options bar.

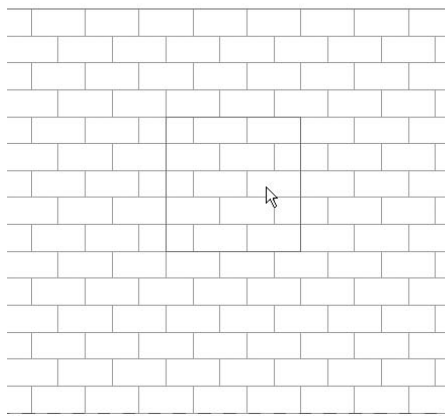
The wall is now shaded and outlined with sketch lines. If you look to the left, you will notice your Design bar has switched to the Sketch tab. This gives you additional functionality as you modify the sketch of the wall (see Figure 6.51).

FIGURE 6.51
You have switched to Sketch mode.



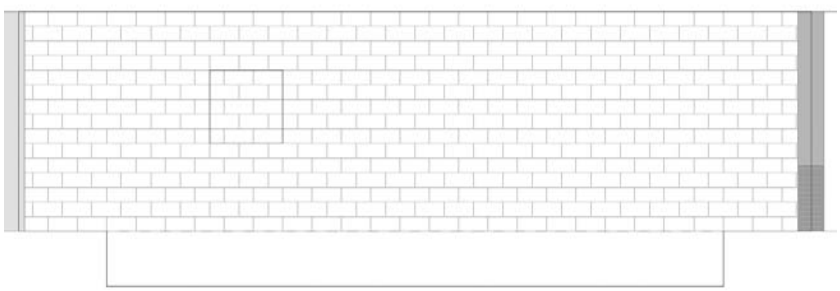
1. Click the Lines button.
2. On the Options bar, you will notice choices similar to when you were placing the walls. You have all of the Sketch options for any shape.
3. Click the Rectangle shape and draw a rectangle in the middle of the wall (see Figure 6.52).

FIGURE 6.52
Draw a rectangle.



4. Now, draw a similar step in the bottom of the wall (see Figure 6.53). You do not need to be accurate for this example.

FIGURE 6.53
Step the bottom of the wall.



5. Click the Finish Sketch button on the Sketch tab. This will end the session and create the new wall profile (see Figure 6.54).

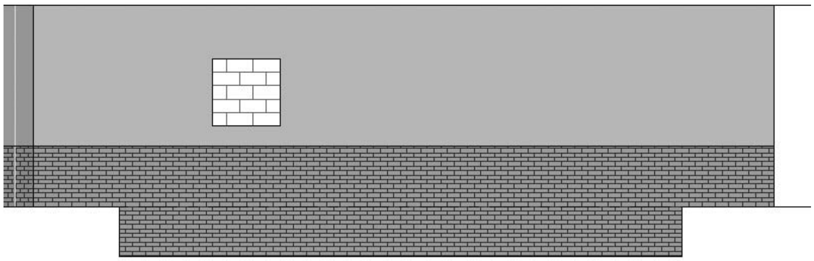


FIGURE 6.54
The new wall profile

A compound wall has its limits. There will be times where parts of a wall do not have a finished face that is aligned. In these situations, you can create a stacked wall instead.

Creating a New Stacked Wall

A stacked wall is exactly what the name implies. It consists of two predefined walls that can be combined and stacked one on another to create a single wall system, where the finished faces are staggered. In the Project Browser, in the Families category, scroll down the list of families until you come to Walls. You will see Basic Wall, Curtain Wall, and Stacked Wall. Expand the Stacked Wall category, and you will see a single Type called Exterior – 8″ Over 10″ Concrete Wall. If you double-click this wall, you will open the Type Properties dialog box. Here you click the Edit button in the Structure row to open the Edit Assembly dialog box, as shown in Figure 6.55.

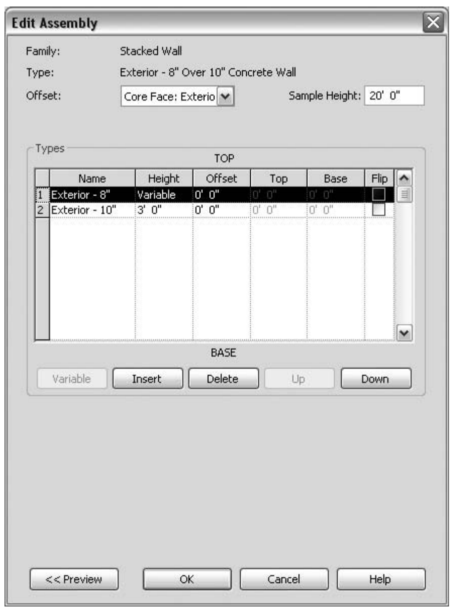


FIGURE 6.55
In the element properties of the stacked wall, you will see that you can edit the assembly.

As you can see, the options differ from those offered in the compound wall's Edit Assembly dialog box. With the compound wall, you are trying to build the wall laterally, layer by layer. With the stacked wall's edit assembly, you are building the wall horizontally, piece by piece. In the Edit Assembly dialog box is a Types field. Here you can stack predefined walls to complete your system. The default stacked wall, for example, consists of a 10″ concrete wall and an 8″ concrete wall. These are two basic walls added to the Types field. To add a new wall to the stack, click on either type 1 or type 2. Below the Types field, you will see an Insert button. Once you click the Insert button, a new wall will be added to the stack. You can move this wall up or down, or leave it right where it is. You can add as many walls to the stack as you need.

In the upper-right corner of the dialog box is a sample height field. Usually when building a stacked wall, it is a good idea to set this to the floor-to-floor height that the wall will be constrained to. (If it is 12′-0″ from Level 1 to Level 2, set this value to 12′-0″.) This helps to avoid unexpected results as you place the wall in the model.

A common issue with walls often involves a pitched roof. You will almost certainly find yourself in a situation where you have a wall that must be extended to meet a roof.

Attaching to a Roof

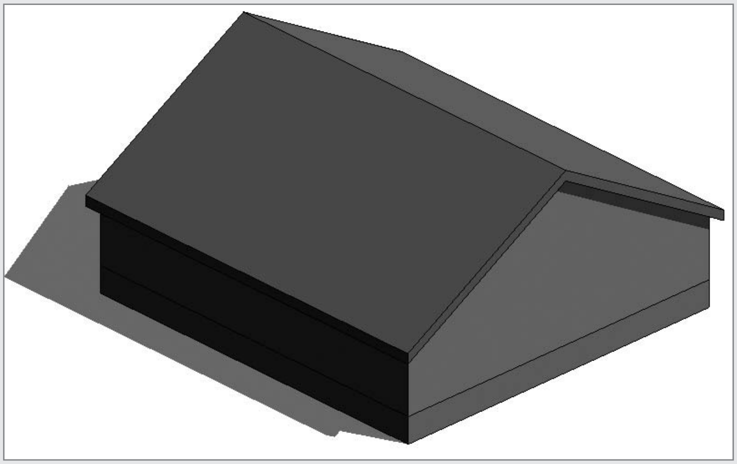
A nice function in Revit Structure is the ability to attach the wall to a sloping or pitched roof system. To do this, you must first draw some walls that define an exterior perimeter. Be sure to constrain the tops of the walls to a level. In the Architectural bar, add a roof to the top of the walls. The roof will have one gable end. Go to a 3D view. If the roof is not visible, press VG on your keyboard. Then, in the Visibility/Graphic Overrides dialog box, turn the display of roofs on. The following exercise explains what you do next.

Exercise: Attaching a Wall to a Roof

To begin, start a new model. You will first need to draw some walls. Then, you will need to go to the highest level and draw a roof as well. For this example, make sure the roof is pitching and has a gable end.

1. Select the wall at the gable end.
2. In the Options bar, you will see an Attach button, next to the words Top/Base. Click this button.
3. The Options bar changes for you to select either a top or a base.
4. Select Top.
5. Now select the roof. The wall will cut itself into the roof, as shown in the following illustration. The walls can be joined to any roof structure. If the roof changes pitch, the walls will change along with it.

6. Repeat the procedure on the other side.



The ability to join walls in Revit Structure to a roof system can be quite helpful (to say the least) when you are trying to support a pitched roof. You must, however, create the roof in Revit Structure—you cannot link a roof from Revit Architecture.

The Bottom Line

Place walls in your model. When you are adding walls to your Revit Structure model, you can rely on the fact that you will get both the width and height of the wall depending on that wall's type, as well as the height constraint you have set. Once the walls are placed in the model, you can easily adjust them to change when the building changes.

Master It Walls are quite simple to place in the model, but they can also be inaccurate if they are not added to the model deliberately. What are three things to look out for when placing walls in the model?

Create new walls. There is a good amount of functionality included in the process of building a new wall type. Furthermore, walls are a system family, which can be used as a basis for any additional wall type you may wish to create.

Master It The 14 walls in the default Revit Structure template are not going to be sufficient. Revit Structure provides the capability to modify a preconfigured wall system to suit your condition. Describe the procedures for:

- Creating a compound wall type
- Creating a stacked wall type

Modify walls in-place. When a wall has been placed, there are some additional functionalities allowed in Revit Structure for the modification of the walls that have been placed.

Master It Walls must conform to various conditions vertically, such as odd openings and a stepped base profile. Also, if you have a pitched roof, the tops of the walls need to be extended to meet the roof. Explain how these procedures can be performed.

Chapter 7

Structural Framing

By now you are starting to get a feel for navigating the Revit Structure interface, and you are ready to delve more deeply into the techniques for modeling a structure. You have studied columns, floors, and decks, and now you will learn how to add different types of framing to your project.

This chapter shows you how to create steel, concrete, and wood framing with the tools supplied by Revit Structure. To be able to model well, you need many tools in your tool chest. If you were out doing carpentry just having a hammer and a saw would not get you very far. So it is with Revit Structure.

This chapter will deal with floor and roof framing as well as moment and braced frames. Of course, as someone working in the structural engineering field, you will find this information critical since you are responsible for putting the "bones" in the building.

The objective for you in this chapter is to learn the techniques required for adding framing members to your working model. When you finish this chapter, you will have a good, working knowledge of how to populate your project with various steel, concrete, or wood framing members.

In this chapter you will learn to:

- Understand structural framing families and properties
- Add floor framing
- Add roof framing
- Create moment and braced frames

Structural Framing Families and Properties

Revit Structure ships with many types of framing families, each of which can be easily loaded into your project. To keep your project from getting too big and slow, you only load in those member sizes that you require for that particular project. The framing libraries encompass all those found in the AISC library, and many international libraries are included as well. You probably realize by now that Revit Structure has a truly global reach, and one that works well with both metric and imperial settings.

Here is a partial list of the extensive framing families available in Revit Structure that you can load into your projects:

- Concrete
  - Concrete: Rectangular Beam
  - Pan Joist
  - Pan Joist with Ledges
  - Precast: Double Tee
  - Precast: Hollow Core Slab
  - Precast: I Shaped Beam
  - Precast: Inverted
  - Precast: L Shaped Beam
  - Precast: Rectangular Beam
  - Precast: Single Tee
  - Precast: Solid Flat Slab
- Light Gauge Steel
  - Light Gauge: Angles
  - Light Gauge: Furring Channels
  - Light Gauge: Furring Hat Channels
  - Light Gauge: Joists
  - Light Gauge: Runner Channels
  - Light Gauge: Zee
- Steel
  - BG Joist Girder
  - Castellated Beam
  - C-Channel
  - Cellular Beam
  - DLH – Series Bar Joist
  - Double C-Channel
  - G Joist Girder
  - GP – Bearing Pile

  - HSS – Hollow Structural Section
  - HSS – Round Structural Tubing
  - K – Series Bar Joist – Angle Web
  - K – Series Bar Joist – Rod Web
  - L-Angle
  - LH-Series Bar Joist
  - LL-Double Angle
  - MC-Miscellaneous Wide Flange
  - MT- Structural Tee
  - Plate
  - Round Bar
  - S-American Standard
  - SLH-Series Double Pitch Joist-5 Panel
  - SLH-Series Double Pitch Joist-7 Panel
  - SLH-Series Double Pitch Joist-9 Panel
  - SLH-Series Double Pitch Joist-11 Panel
  - SLH-Series Double Chord Bar Joist
  - ST-Structural Tee
  - VG Joist Girder
  - WRF-Welded Reduced Flange
  - WT-Structural Tee
  - WWF-Welded Wide Flange
  - W-Wide Flange
- Wood
  - Dimensional Lumber
  - Glulam-Southern Pine
  - Glulam-Western Species
  - LVL-Laminated Veneer Lumber
  - Open Web Joist

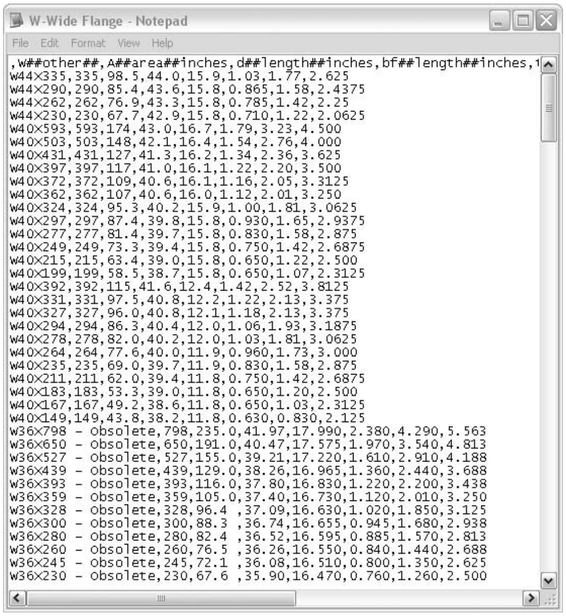
As you can see, there are many types and varieties of members you can use to make your virtual model.

Working with the Framing Libraries

But what if the framing member you require for your project does not exist in the Revit libraries? When you first start modeling, you will rely heavily on the built-in libraries. If what you want is not there, you can check for families at the websites for AUGI (Autodesk Users Group International) or Revit City, where many people have uploaded their own families for others to use. If you still cannot find what you want, you will have to build your own. After you have had a chance to try it a few times you will find the process of adapting and creating your own families is not that hard and that it is not as intimidating as it sounds.

If a member size or shape does not exist in the Revit libraries, you have several alternatives. You can create your own structural framing member from scratch using solid modeling tools, or you can adapt one of the existing libraries to fit your needs. You adapt an existing library by editing the text database file that is associated with it, in order to add odd member sizes or out-of-date sizes (see Figure 7.1) that might not already be there.

The text file is located in the Framing folder along with the Revit family file (with the `.rfa` extension). The text file has the parameters listed on the top line that describe the basic shape of the element, such as flange width (bf) and web thickness (tw) that you will have to provide in order to make your new size work.




```
,W##other##,A##area##inches,d##length##inches,bf##length##inches,
W44x335,335,98.5,44.0,15.9,1.03,1.77,2.625
W44x290,290,85.4,43.6,15.8,0.865,1.58,2.4375
W44x262,262,76.9,43.3,15.8,0.785,1.42,2.25
W44x230,230,67.7,42.9,15.8,0.710,1.22,2.0625
W40x593,593,174,43.0,16.7,1.79,3.23,4.500
W40x503,503,148,42.1,16.4,1.54,2.76,4.000
W40x431,431,127,41.3,16.2,1.34,2.36,3.625
W40x397,397,117,41.0,16.1,1.22,2.20,3.500
W40x372,372,109,40.6,16.1,1.16,2.05,3.3125
W40x362,362,107,40.6,16.0,1.12,2.01,3.250
W40x324,324,95.3,40.2,15.9,1.00,1.81,3.0625
W40x297,297,87.4,39.8,15.8,0.930,1.65,2.9375
W40x277,277,81.4,39.7,15.8,0.830,1.58,2.875
W40x249,249,73.3,39.4,15.8,0.750,1.42,2.6875
W40x215,215,63.4,39.0,15.8,0.650,1.22,2.500
W40x199,199,58.5,38.7,15.8,0.650,1.07,2.3125
W40x392,392,115,41.6,12.4,1.42,2.52,3.8125
W40x331,331,97.5,40.8,12.2,1.22,2.13,3.375
W40x327,327,96.0,40.8,12.1,1.18,2.13,3.375
W40x294,294,86.3,40.4,12.0,1.06,1.93,3.1875
W40x278,278,82.0,40.2,12.0,1.03,1.81,3.0625
W40x264,264,77.6,40.0,11.9,0.960,1.73,3.000
W40x235,235,69.0,39.7,11.9,0.830,1.58,2.875
W40x211,211,62.0,39.4,11.8,0.750,1.42,2.6875
W40x183,183,53.3,39.0,11.8,0.650,1.20,2.500
W40x167,167,49.2,38.6,11.8,0.650,1.03,2.3125
W40x149,149,43.8,38.2,11.8,0.630,0.830,2.125
W36x798 - Obsolete,798,235.0,41.97,17.990,2.380,4.290,5.563
W36x650 - Obsolete,650,191.0,40.47,17.575,1.970,3.540,4.813
W36x527 - Obsolete,527,155.0,39.21,17.220,1.610,2.910,4.188
W36x439 - Obsolete,439,129.0,38.26,16.965,1.360,2.440,3.688
W36x393 - Obsolete,393,116.0,37.80,16.830,1.220,2.200,3.438
W36x359 - Obsolete,359,105.0,37.40,16.730,1.120,2.010,3.250
W36x328 - Obsolete,328,96.4 ,37.09,16.630,1.020,1.850,3.125
W36x300 - Obsolete,300,88.3 ,36.74,16.655,0.945,1.680,2.938
W36x280 - Obsolete,280,82.4 ,36.52,16.595,0.885,1.570,2.813
W36x260 - Obsolete,260,76.5 ,36.26,16.550,0.840,1.440,2.688
W36x245 - Obsolete,245,72.1 ,36.08,16.510,0.800,1.350,2.625
W36x230 - Obsolete,230,67.6 ,35.90,16.470,0.760,1.260,2.500
```

FIGURE 7.1
The text database file for wide flange beams can be easily edited.

To add a new size to the existing family file, first add a description of the element, such as **`W12x16.5 – Obsolete`**. Then fill in the values that describe its shape, as in the following parameter list for a steel beam with imperial units:

- **W**##other##
- **A**##area##inches

- **d**##length##inches
- **bf**##length##inches
- **tw**##length##inches
- **tf**##length##inches
- **k**##length##inches

Make sure each parameter is comma delimited (with no spaces) and follow this order exactly. So to add a new shape to an existing library, simply add a new line anywhere in the text file and fill in the values you want for the various parameters that determine the shape of the element. When you re-load the family into your project the new size you created will be available for placement into your project. That is one form of adaptation for an existing framing family.

Another example of that sort of family adaptation is one that you will see in Chapter 19, where you add a wood nailer element to the top of a steel beam family for use in a project that has both wood and steel framing (which is a common occurrence for many commercial facilities).

Floor and Roof Deck Constraints

For the most part, the framing that you add to your building project, whether it is steel, concrete, or wood, will be supporting some type of elevated deck (see Figures 7.2, 7.3, and 7.4). Of course, in some projects the majority, or even all, of the framing supports piping or mechanical devices such as industrial production facilities for gas and electric plants, or test frames for new aircraft.

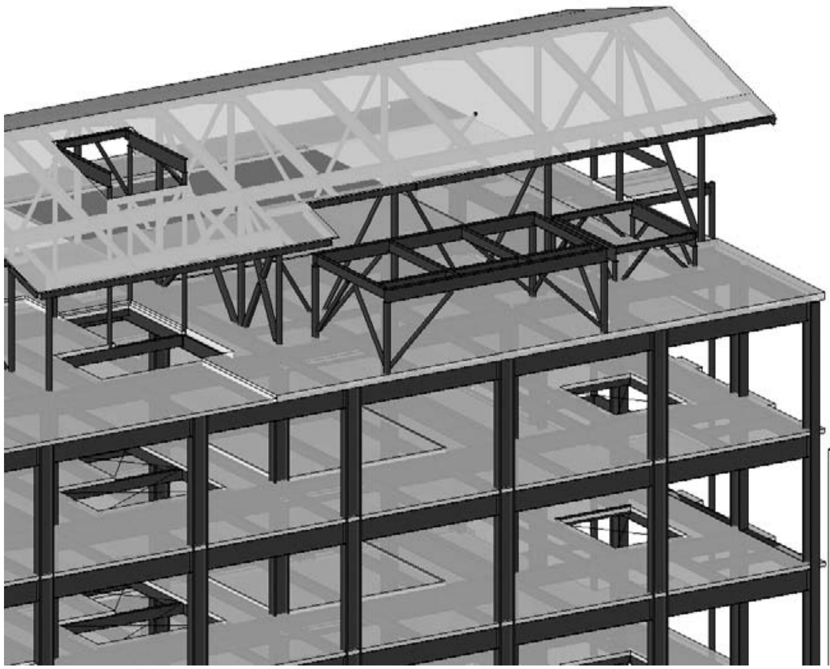


Image courtesy of Brandow & Johnston Inc.

FIGURE 7.2
Steel-framed office structure with composite metal deck and concrete floors with a sloping roof to support Spanish tile, and mechanical equipment platforms for HVAC

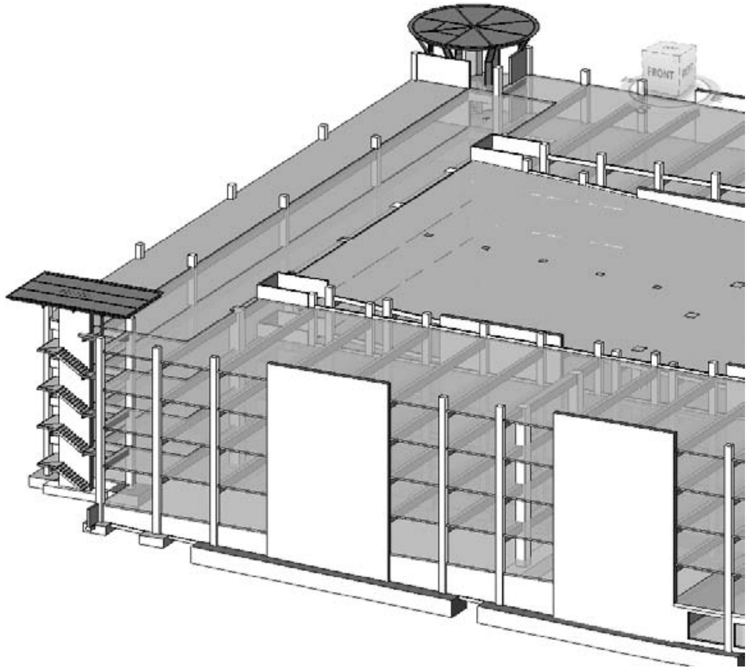


Figure 7.3
Concrete parking structure with prestressed, pretensioned (PT) concrete framing supporting PT decks

Image courtesy of Brandow & Johnston Inc.

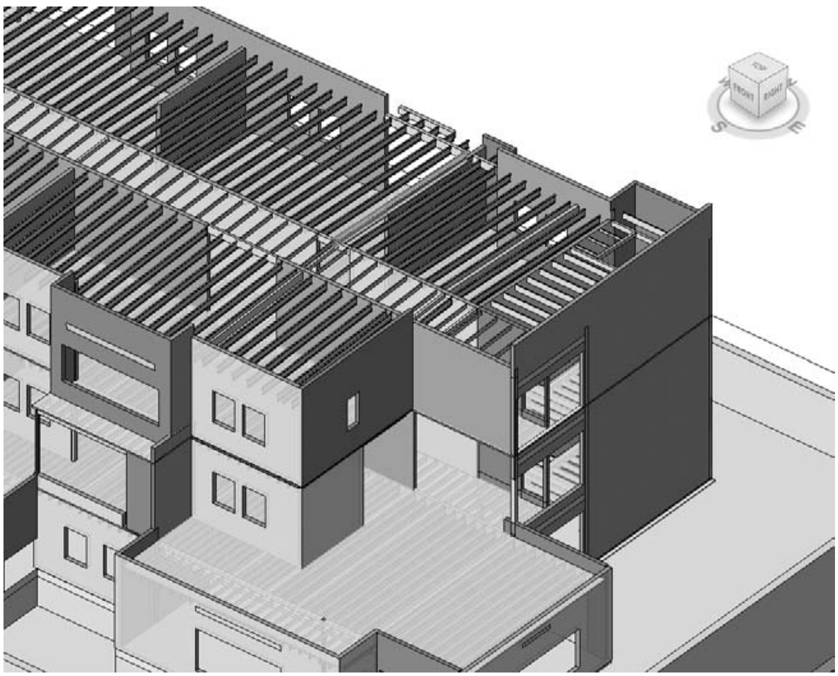


Figure 7.4
Multiple housing structure framed with lumber connecting to wood bearing and shear walls

Image courtesy of Brandow & Johnston Inc.

One important consideration when adding framing elements to your project is establishing the reference plane to which they are attached. This issue goes to the very core of Revit Structure and its constraint-based philosophy. Since you are working through an ever-changing design process, from schematics through to construction documentation, the model must always have as much flexibility as possible. You need this flexibility in your model to do such things as stretch to different-sized bay widths and to be able to adjust story-to-story heights between levels in case the design changes.

The basic vertical constraint for floor framing in your project will be the level to which the framing is attached. The basic horizontal constraints will be your grid system. Constraining your framing elements to these two datum elements will allow you to "flex" the model when needed as the design process continues.

The level you draw the framing on is the default for its constraint, so if the level moves, the framing will move with it. One added complication is that the framing is often below the deck, so members need to be offset downward below the deck element. You can do this by highlighting the member and changing the values for its offsets in the Element Properties dialog box. It is easy to forget to do that and you end up with framing at the top of the deck unless you are paying close attention. Also, if the deck depth changes due to design updates, you have to remember to select and edit all the framing member offsets. So what is the best practice?

Many advanced users elect to use the bottom of the deck as the reference plane for attaching the framing. That way, they keep the deck constrained to the level, and do not have to worry if the depth of the deck changes. Others create a new level, or use a reference plane at the top of steel elevation so they do not have to worry about setting offsets. That seems like a better way to go.

In the next section, you will learn to create basic floor framing for an elevated deck in your virtual model. As opposed to roof framing on a sloped or warped deck, floor framing is mostly flat and fairly straightforward to model, so that is a good place for you to start in order to understand how this all works.

Developing Good Modeling Skills and Workflow

One efficient way to develop your modeling skills is to have a good understanding of the way buildings are constructed. Go out to any construction site and watch how they work. Study the chronological sequence of adding elements as the building is constructed. Which elements must be constructed first? Columns must be added before girders. Girders must be added before bays can be in-filled, and so on.

In the virtual building world, the workflow is not much different. For most areas of modeling, creating your virtual model similar to how it will be sequenced in the field is a good approach.

Foundations are one virtual element that is in many cases modeled out of sequence to the actual construction process. During the design process, the foundation information usually comes after the analysis of the structure is developed. So at first your foundations may be placeholders that approximate the size and material system. Or they may not be there yet. After all, the model is virtual and won't fall down.

Adding Floor Framing

In this section, you will study the methods used to frame various elevated floor and roof decks. Some of the issues regarding decks have been discussed earlier in Chapter 5. Here are some of the types of roof shapes that you will face when you start modeling that will be explored more fully in this chapter:

- Flat diaphragms for floors
- Sloped diaphragms for roof structures
- Warped diaphragms for roof structures

You will learn to create your framing in a workflow that is similar to how the members would actually be constructed in the field. Assuming that the grids and levels are established in your project, and that columns and decks have been created, now you will move on to the addition of your floor framing. So where do you begin?

How do you proceed when you want to add floor framing to your project? Depending on the way you prefer to work, you can either add the floor deck element first or the framing elements first. But since the framing supports the floor deck layout, it is best practice to proceed with the deck first, though it is not absolutely necessary. That way you can easily follow the shape of the floor while you add its support framing. However, for sloping or warped roof deck framing, it will be essential to create the deck first for reasons that will be explained later in the chapter.

The columns are there already, so the first step to framing the floor will be to connect the columns with girders as you construct your virtual model. You will find the Beam command on the Modeling tab, but first it is important to understand its available options.

The Beam Options Bar Settings

Let's look at the various options now, from left to right, on the Options bar. When you start the Beam command, the Type Selector will list, in a pull-down menu, all the currently loaded beam sizes (see Figure 7.5). Next to the Type Selector is the Beam Element Properties button, where you can set or edit the various parameters of the framing. (See the next section for a discussion of the element properties.) The next button to the right is the Load button, which allows you to easily load new shapes from the framing libraries that you want to use in your project. To the right of that button are the draw and pick buttons, which enable you to add the beam to the view by drawing or picking.

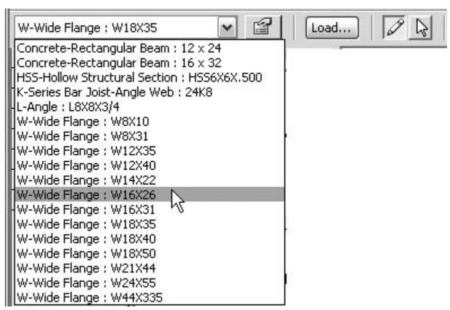
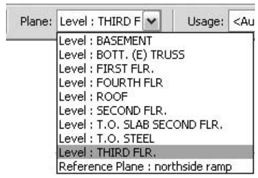


FIGURE 7.5
The Type Selector lets you choose shapes to insert.

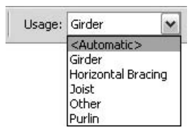
The next pull-down menu (see Figure 7.6) allows you to choose the plane on which you want to add the beam element. Usually the value is set to the floor you are working on, but you can also add it to another plane of your choosing.

FIGURE 7.6
You define the plane on which you want to add your framing.



The next pull-down menu sets the structural usage field for the new beam (see Figure 7.7). As we discussed earlier, there are five categories you can use. If you leave the value on <Automatic>, Revit Structure makes its own judgment as to what structural usage the new element should be assigned. But basically Revit will assign them like this: girders connect to columns, joists connect to girders, and purlins connect to joists. If you leave the structural usage set to <Automatic>, you need to pay close attention because improperly connected beams, or ones where Revit gets confused (yes, Revit does get confused sometimes) will be put into the Other category. The "Other" line weight is usually set to a pen weight of 1 so that those members, when printed, will look very thin.

FIGURE 7.7
The Usage pull-down is a convenient way to preset the beam's usage.

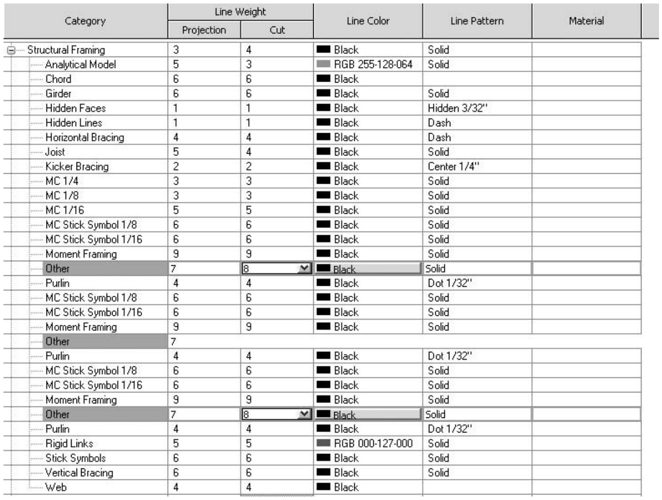


Some people like to set each structural usage type to a different line weight, with the girders being the thickest, in order to distinguish between them. Others set all the lineweights the same weight regardless of structural usage assignments. It is one of those judgments that you need to make for yourself and your own firm.

For each of these beam categories, the line weight and line style properties can be independently defined, so you can make the girders a heavier line weight than joists, or alter the display in whatever way is required for your project. These changes are made by using the Object Styles dialog box (see Figure 7.8).

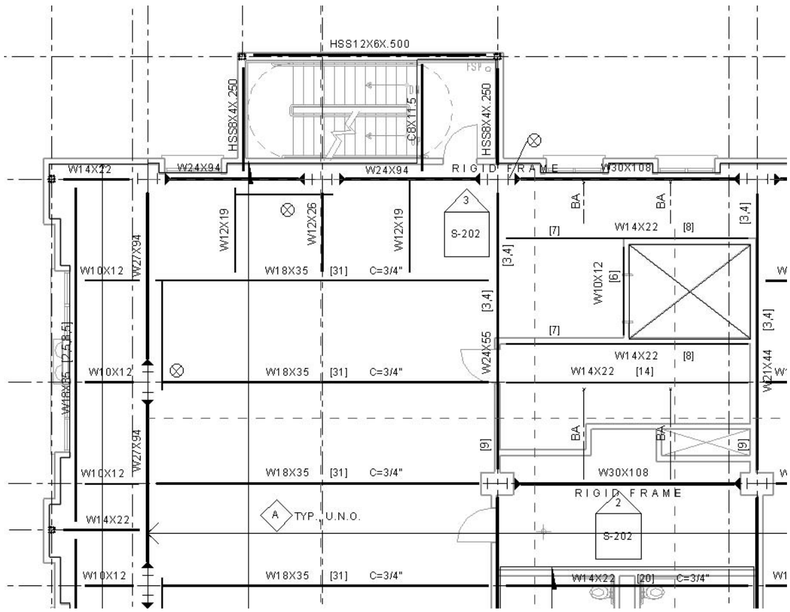
For a good example of how beam presentation can differ, see Figure 7.9. On a structural framing plan, the seismic system (the moment or braced frames) is usually displayed with the heaviest line weight, so it "jumps right out at you," so to speak. The plan view in the figure has the rigid frame member's structural usage parameter set to Other. The Other parameter is then set to a line weight of 7 in the Object Styles dialog box, for projected lines. The girders are set to line weight 6, and the joists are set to line weight 5.

Figure 7.8
The Object Styles dialog box lets you control the display of your beams and girders in their various views. You can specify settings for an individual view or globally.



Category	Line Weight: Projection	Line Weight: Cut	Line Color	Line Pattern	Material
Structural Framing	3	4	Black	Solid	
Analytical Model	5	3	RGB 255-128-064	Solid	
Chord	6	6	Black		
Girder	6	6	Black	Solid	
Hidden Faces	1	1	Black	Hidden 3/32"	
Hidden Lines	1	1	Black	Dash	
Horizontal Bracing	4	4	Black	Dash	
Joist	5	4	Black	Solid	
Kicker Bracing	2	2	Black	Center 1/4"	
MC 1/4	3	3	Black	Solid	
MC 1/8	3	3	Black	Solid	
MC 1/16	5	5	Black	Solid	
MC Stick Symbol 1/8	6	6	Black	Solid	
MC Stick Symbol 1/16	6	6	Black	Solid	
Moment Framing	9	9	Black	Solid	
Other	7	8	Black	Solid	
Purlin	4	4	Black	Dot 1/32"	
MC Stick Symbol 1/8	6	6	Black	Solid	
MC Stick Symbol 1/16	6	6	Black	Solid	
Moment Framing	9	9	Black	Solid	
Other	7				
Purlin	4	4	Black	Dot 1/32"	
MC Stick Symbol 1/8	6	6	Black	Solid	
MC Stick Symbol 1/16	6	6	Black	Solid	
Moment Framing	9	9	Black	Solid	
Other	7	8	Black	Solid	
Purlin	4	4	Black	Dot 1/32"	
Rigid Links	5	5	RGB 000-127-000	Solid	
Stick Symbols	6	6	Black	Solid	
Vertical Bracing	6	6	Black	Solid	
Web	4	4	Black		

Figure 7.9
A typical floor framing plan for a steel building with differing line weights for beam elements per usage



To the right of the Structural Usage pull-down, you have the Grid options button (see Figure 7.10), which allows you to automatically place girders on grid lines by simply selecting them. To the right of that button is the Tag check box, where you can automatically add tags to the element after placement. Since tagging is only displayed in the view in which it is created, you might not want to have tags in a working view, so you can deselect this check box before placement if you want to. To the right of the Tag check box is the 3D Snapping check box. If you select this check box, the beam will snap to the elevation of the intersecting beam or girder. This is an especially important option when you're doing roof framing. When working with sloping roof members and when framing to tops of columns that are at different elevations, you have no

plane on which to place the beam. So having Rivet Structure automatically find the intersecting point is critical to being able to easily and efficiently place the framing member.

The Chain option, when checked, allows you to frame from point to point to point without stopping. Finally, at the very right of the Beam Options bar are the drawing tools, which you can use if you have enabled the Draw command.

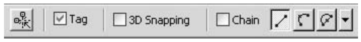


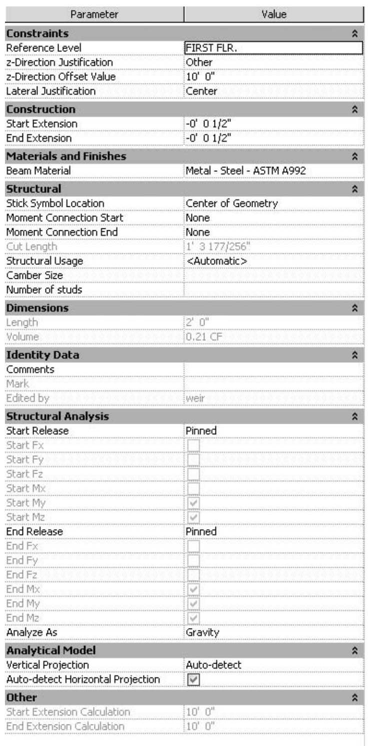
FIGURE 7.10
On the right side of the Options bar you will find placement options.

Beam Element Properties

Next to the Type Selector on the Options bar is the Element Properties button. You set or edit the various parameters of the framing element there (see Figure 7.11). You will be going in and out of there constantly to adjust the properties of your elements. The dialog box includes the following major areas: Constraints, Construction, Materials and Finishes, Structural, Dimensions, Identity Data, Structural Analysis, Analytical Model, and Other. Each of those categories has a set of parameters that are related. If you highlight an existing beam and right-click, you can also open the Element Properties dialog box by selecting Element Properties at the bottom of the pop-up menu.

You can set some of the element instance parameters ahead of time before placement which makes multiple insertions easier and more efficient so you do not have to repeatedly edit their properties after placement in the model. Be careful when presetting the parameters, though, and remember to change them at the appropriate time.

FIGURE 7.11
The Element Properties dialog box instance parameters



Parameter	Value
Constraints	
Reference Level	FIRST FLR.
z-Direction Justification	Other
z-Direction Offset Value	10' 0"
Lateral Justification	Center
Construction	
Start Extension	-0' 0 1/2"
End Extension	-0' 0 1/2"
Materials and Finishes	
Beam Material	Metal - Steel - ASTM A992
Structural	
Stick Symbol Location	Center of Geometry
Moment Connection Start	None
Moment Connection End	None
Cut Length	1' 3 177/256"
Structural Usage	<Automatic>
Camber Size	
Number of studs	
Dimensions	
Length	2' 0"
Volume	0.21 CF
Identity Data	
Comments	
Mark	
Edited by	weir
Structural Analysis	
Start Release	Pinned
Start Fx	☐
Start Fy	☐
Start Fz	☐
Start Mx	☐
Start My	☑
Start Mz	☑
End Release	Pinned
End Fx	☐
End Fy	☐
End Fz	☐
End Mx	☑
End My	☑
End Mz	☑
Analyze As	Gravity
Analytical Model	
Vertical Projection	Auto-detect
Auto-detect Horizontal Projection	☑
Other	
Start Extension Calculation	10' 0"
End Extension Calculation	10' 0"

Adding Floor Framing to Your Virtual Model

Now that you have studied the various options for beams, their placement, and for their instance parameters, you are ready to start adding framing elements to your project. Begin with girder placements from column to column.

To add girders to your project one at a time:

1. Go to the view where you want to add the framing.
2. On the Modeling menu, click the Beam button.
3. In the Type Selector, choose a size to add, or load one from the framing libraries.
4. On the Options bar, select a Usage type for the beam you are adding, or leave it set to Automatic.
5. For floor framing make sure 3D Snapping is unchecked.
6. Select the Chain check box to easily create girders from column to column in a continuous pattern (or "chain").
7. Select the Tag box on the Options bar to automatically add the size of the beam above the framing member once it is placed.
8. Hover over a column, and use the Tab key to cycle through the various snaps until the appropriate snapping icon appears. Use the midpoint when possible.
9. Click to pin one end of the girder onto the column.
10. Click at the next column to finish placing the beam.

There are some time-saving techniques you will want to use when adding girders to your project. You can add girders by snapping from column to column individually as just described, and use the Chain option for continuous placement. Or you can use the Grid option, which allows you to add girders between columns automatically, simply by selecting grid lines (see Figure 7.12).

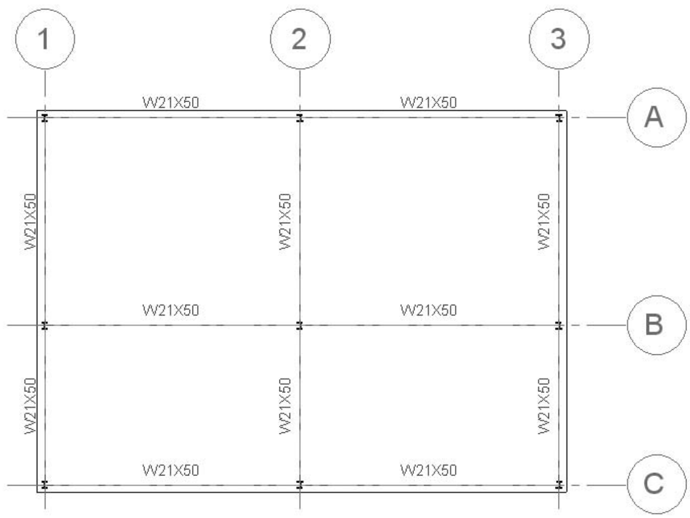


FIGURE 7.12 The Grid option allows for easy girder placement between columns. As shown, the members are ghosted in between columns with all grids chosen.

Place beam elements by clicking the grid:

1. Perform Steps 1 through 5 from the previous task in the same way.
2. Click the Grid button on the Options bar to use the grid method.
3. Highlight a grid that has columns you want to frame between.
4. Rivet Structure will add girders between columns on that grid. The girders will appear ghosted until you finish.
5. Hold the Ctrl key down to select multiple grids at one time.
6. When you are done selecting, click Finish on the Options bar to complete placement.

After finishing the placement of the girders between the columns on your floor plan, you are ready to in-fill those bays with beams. After that, you can add secondary members for floor openings, stair framing, and other miscellaneous framing elements, drawing them individually. This chronology is another example of how the workflow for virtual modeling is similar to the way it would be erected in the field.

Modeling Is Half the Battle

Creating the model is only half of the battle. The other half is to display and derive the two dimensional plans, sections, and elevations that comprise your final construction documents set. In the current discussion on beams, that explains why you must pay close attention to the structural Usage settings so your framing displays accordingly. Of course, you could set all of the beam weights for the various usages at a single line weight and ignore the usage if you want to keep things simple. Display issues are constantly occurring, so you must understand how to use the display tools you will need in order to adjust the display accordingly.

In-filling Bays with the Beam System Tool

Now that the framing bays are bounded by girders, you use the Beam System tool, found on the Modeling tab, to add the beams that will fill the framing bays (see Figure 7.13). Beam System is a powerful and versatile tool, and one that you will use on most projects. Its benefits lie in its ease of use and its ability to “flex” when or if the framing bay dimensions change.

The basic approach in using this tool consists of:

- Defining the area within which the beams will be drawn
- Defining the direction of the beam elements
- Establishing layout rules for their placement

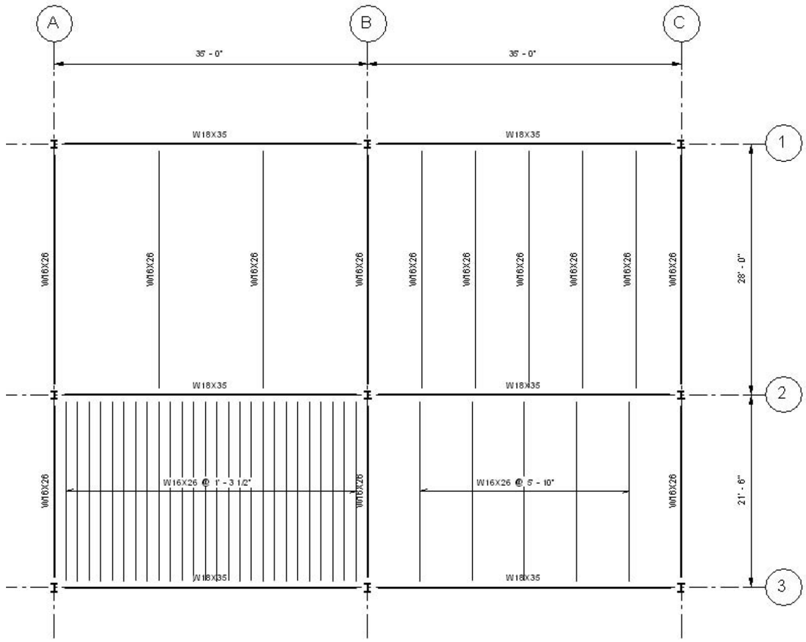


FIGURE 7.13 Examples of in-filled framing bays using different layout rules and tag types

There are two basic approaches to defining the area and direction of the beams: the one-touch method and the sketch method. In the one-touch method you simply hover over one of the girders in the direction you wish the framing to be placed in the bay, then click to finish. With the Sketch method you sketch lines that enclose the area of the in-fill. This method will work when the one-touch method fails. Let's take a look at the Options bar to learn the various options available for the Beam System command.

On the left side of the bar (see Figure 7.14) is the Type Selector pull-down which allows you to select different types of beam systems that you might have created. Next to that is the Beam System Properties button. The Beam Type pull-down is next and allows you to select the beam type for the layout. Only one beam type can be called out for any one placement of the beam system. Then there is the Sketch button, which we will discuss in a moment.

FIGURE 7.14 The Options bar for a beam system

Four options are available when you're placing beam elements (see Figure 7.15):

- The Clear Spacing option sets the distance between members from the edge of the flange to the edge of the adjoining flange rather than at the centerline of the elements.
- The Fixed Distance and Fixed Number options do just what you'd expect.

- Finally, the Maximum Spacing option in-fills the beams using the value in the box to the right of the Layout box. This is the most powerful option because it remains dynamic if the bay dimensions change. Over the course of the project, if the bay expands enough so that the members exceed the Maximum Spacing value, then new members will automatically be added into the bay. If the bays get smaller, members will be eliminated. This helps you achieve your goal of making the model as flexible as possible during the design process.

Figure 7.15
The four Layout options

The Justification parameter on the Options bar (see Figure 7.16) controls how the beams are spaced within the bay. The spacing can start from either side, or start from the center of the bay.

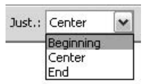


Figure 7.16
The Justification option

Checking the 3D option (see Figure 7.17) helps you place beams when the member it intersects is sloping, You will be hearing a lot about this one later on in the chapter. It allows you to place a beam and have it snap to an intersecting member, such as a sloping roof girder.

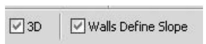


Figure 7.17
The 3D option

The Tag option (see Figure 7.18), if checked, will enable two types of beam tags: an overall bay type or individual callouts. The overall type is best used for framing, such as wood-framed floors with many members where spacing is very tight. If you tried to tag each one of those elements, it would look too crowded on your plan.

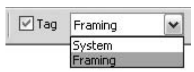


Figure 7.18
The Tag option

As mentioned earlier in this section, there are two methods of placement: one-touch and sketch. The one-touch method is easy to use (see Figure 7.19): you simply hover over a beam in the direction you want the framing to be added. The framing then will ghost in with blue dashed lines until you click for final placement. You will find that this method allows you to complete a whole floor of framing on your model in minutes. It is so easy that sometimes it is faster to erase the framing and redo it if design changes warrant it. Remember, though, that the one-touch method is not available as an option unless you have a bay enclosed with beam elements.

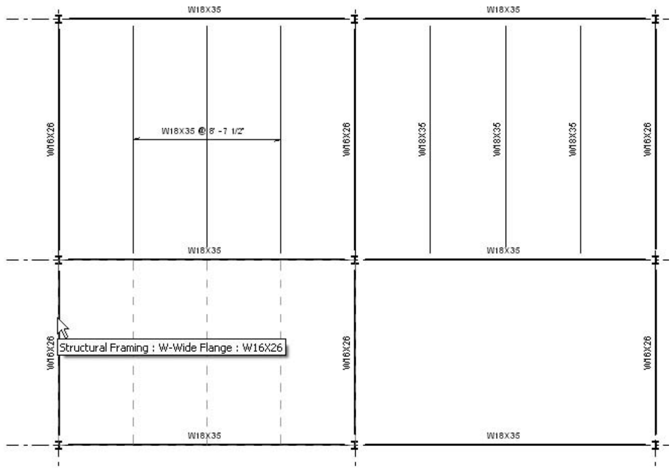


FIGURE 7.19
The beam system one-touch method

In some cases, the one-touch method does not work, and so you must use the sketch method. First, click the Sketch button on the Options bar (see Figure 7.20). That enables Sketch mode on the Design bar. By clicking Lines in the Sketch panel, you enable the drawing tools on the Options bar so that you can sketch the lines that define the shape you require for that bay (see Figure 7.21). Or you can use the Pick Supports option to select the elements that form the boundary of the framing bay. You can also control work plane attributes in this dialog box.

An important option in this menu is the Reset Structural Beam System function. If after placement your design criteria has changed and you must re-space the elements in the bay, you will find that sometimes the display does not update correctly to the new bay shape and still displays the stick symbols as they were. This is especially true when you're expanding a bay. The members might not display to the new extents. In that case, you need to enable this option to make the display appear correctly.

So that explains the various properties and options available for you to use in the placement of beams in your project. Next you will take a look at how to add curved beam framing to your model.

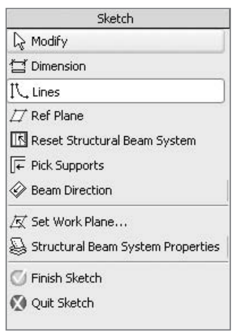


FIGURE 7.20
The beam system Sketch menu on the Design bar

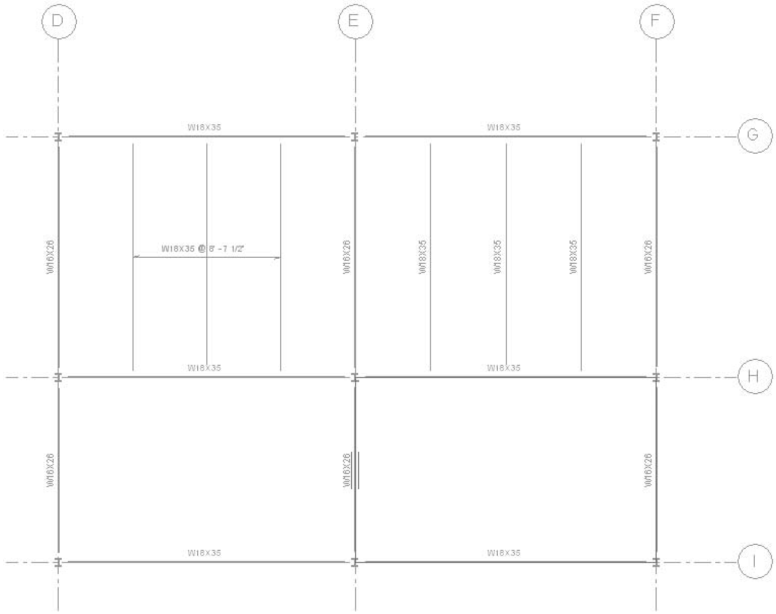


FIGURE 7.21
Sketching in the beam system bounded area

Using Curved Beams

Almost every building project has a curved beam or two (see Figure 7.22), so it is very important to be able to easily create them in your model. For example, you might need one in order to create support for a barrel-style roof, or for supporting an exterior curved edge of a deck. You can create curved beams using the drawing tools on the Options bar. You can draw such shapes as an arc, circle, or spline as a path for the beam shape. You create curved beams as well by picking a curved line on your plan.

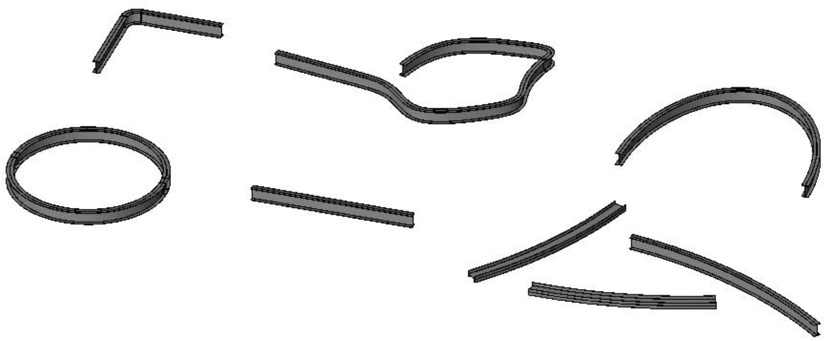


FIGURE 7.22
Some curved beam shapes drawn with straight, circular, arc, and spline options

To create a curved beam on your floor plan, do the following:

1. On the Modeling bar, click Beam.
2. In the Type Selector, set the beam shape you want to use, or load a new beam shape from the framing libraries.
3. On the Options bar, click one of the drawing tools such as a line or an arc, and draw the centerline of the member on the plan. Or you can click the Pick button and select a line in your project, and the curved beam will follow it.

Other Important Types of Floor Framing

You might have wondered in Chapter 5 why we didn't discuss concrete pan joist systems (waffle slabs) or precast slab systems. The reason is that these subjects seem better dealt with here, since the concrete slab and concrete framing are monolithically constructed. As opposed to the approach we took for steel and concrete beams, these systems load in as units for specific layouts.

Pan Joist Supported Slabs A pan joist system can be easily constructed by using the Pan Joist concrete framing family. You create the waffle slab by placing a beam system layout in each direction in the same bay (see Figure 7.23). The two beam system placements will "clean up" and display monolithically.

Precast Supported Slabs Precast hollow-core slabs, Single, and Double-T system families are all available in the Revit Structure concrete framing library (see Figure 7.24). Individual panels are created to specific widths and then arrayed across the area of the floor. The framing libraries contain the basic framing shapes you will need for precast framing, with Precast Inverted-T girders, L-Shaped girders, and more.

Figure 7.25 shows some sections that have been cut from a precast structure. 2D annotation is added, as well as reinforcing. The reinforcing can be either added as a 3D component of the element or in 2D as drafting detail lines.

Prestressed and Post-tensioned (PT) Framed Concrete Structures Concrete structures using PT (prestressed post-tensioned) systems are quite common and can be readily modeled in Revit Structure (see Figure 7.26). PT beams are nonrectangular, so you can use the Pan Joist family or adapt your own family depending on how accurate you want the beam to appear.

It is important for you to develop good judgment about the extent to which you add detail in your model families. Adding in a 1″ chamfer to the concrete beam interface with the slab might be accurate, but that might be unnecessary if you do not intend to use views in your construction documents for a lot of detailing.

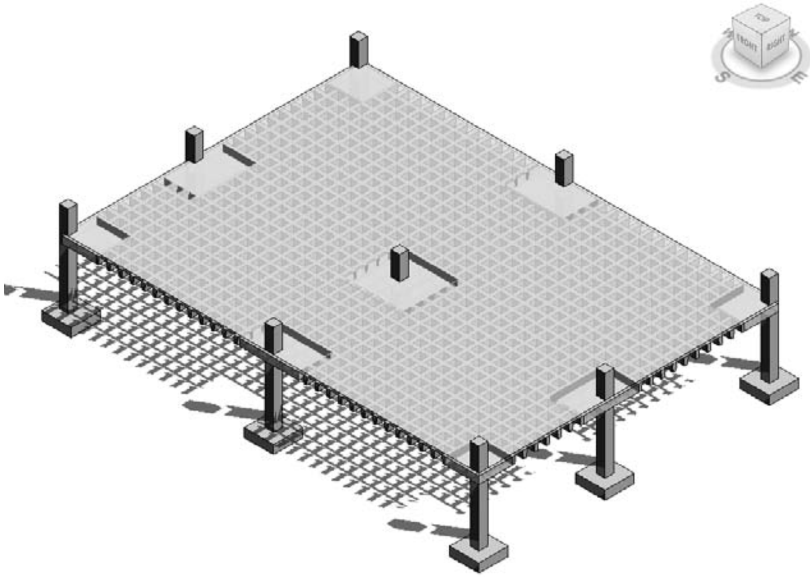


FIGURE 7.23
Concrete pan joist system with filled cores at the columns

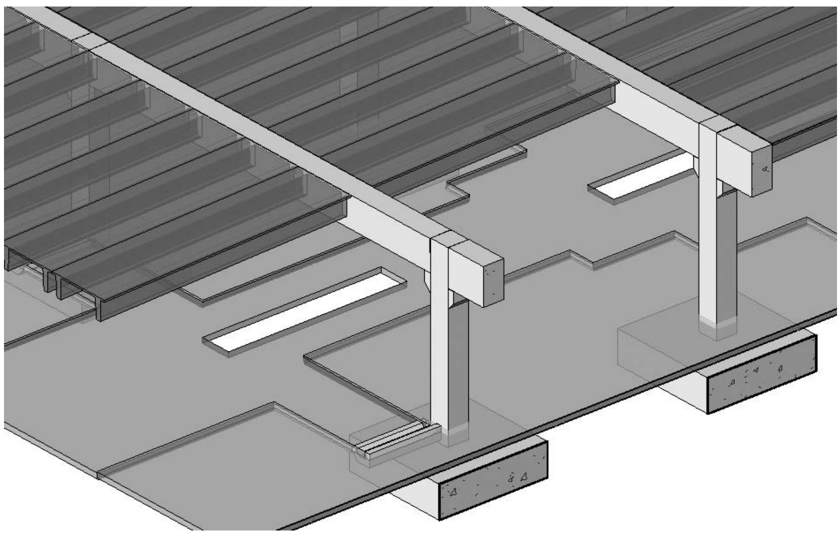


FIGURE 7.24
Many framing types are available in Revit Structure, like this precast concrete system.

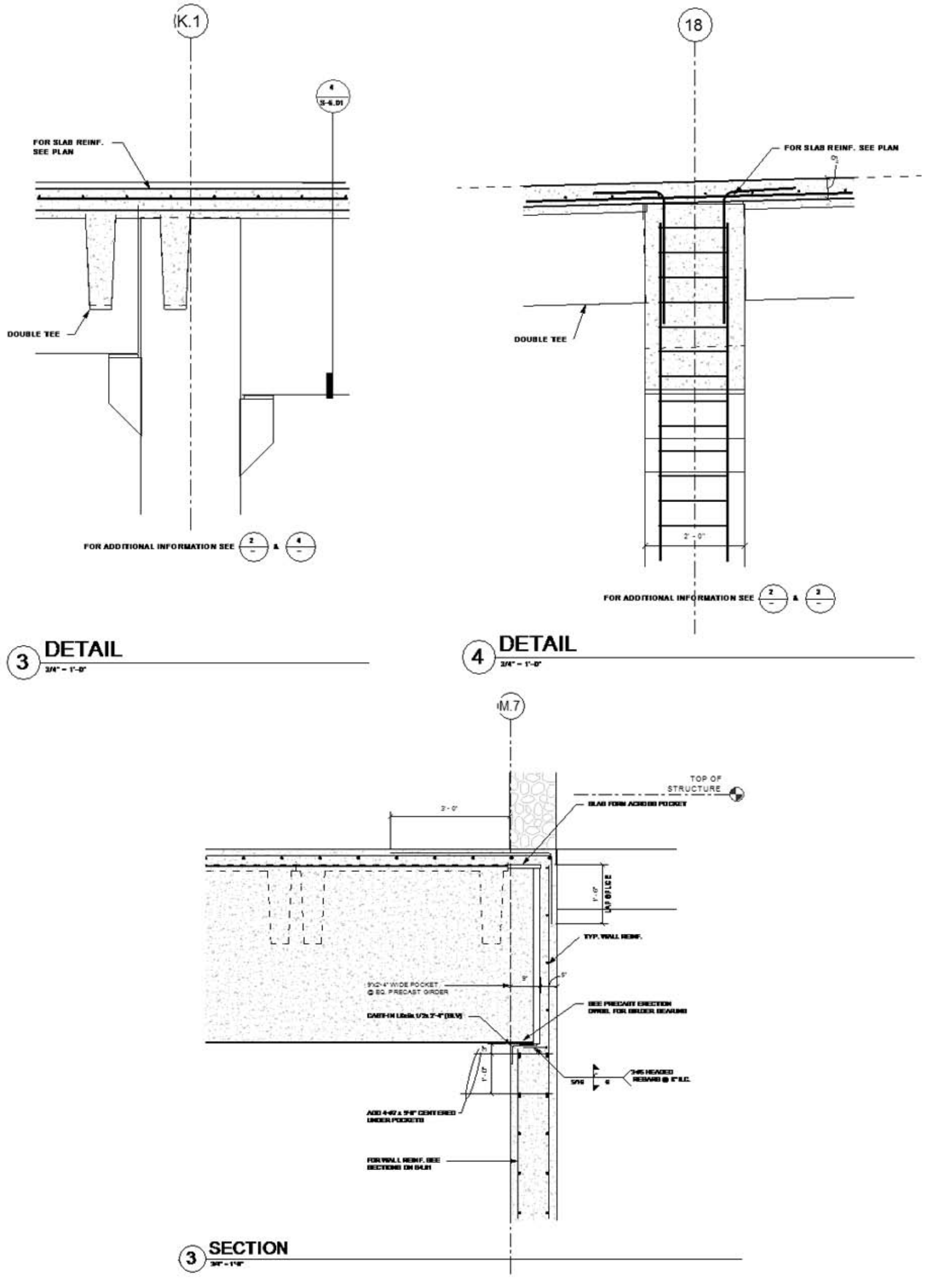


FIGURE 7.25 Sections and details for a precast concrete framing project

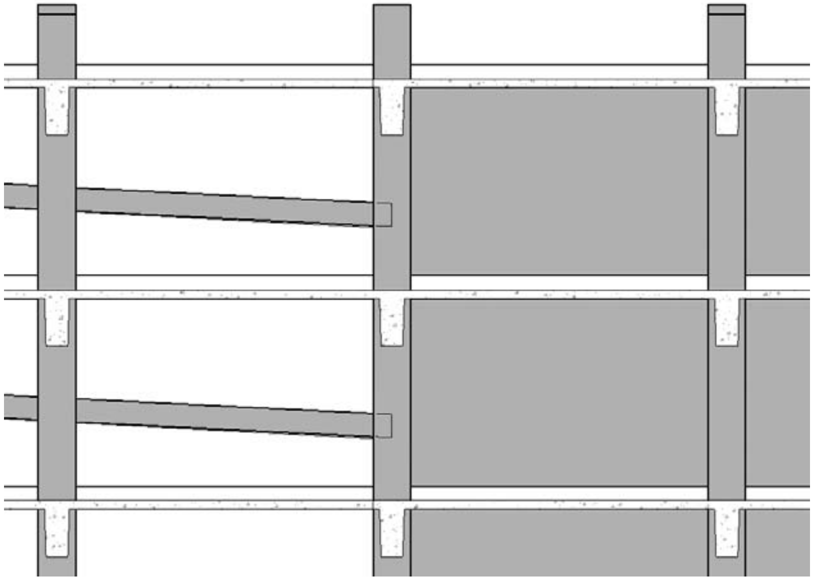


FIGURE 7.26
Elevation at ramp and shear wall for a concrete PT parking structure

In the next exercise, you will practice placing in-fill beams into a project using various methods of placement.

EXERCISE: ADDING GIRDERS TO A FLOOR PLAN

This exercise will have you add girders first and then in-fill beams on several bays in a simple steel structure. You'd use the same procedure for concrete or wood framing members.

First, you place the girders individually, then by the grid method:

1. Open `Dataset_0701_begin`, where you should see a grid and steel columns.
2. On the Design bar, click the Modeling tab and then click Beam.
3. Select a W18x35 beam type in the Type Selector.
4. On the Options bar, select Girder in the Structural Usage field, and select the Chain check box.
5. Hover over the column at Grid A1 until a red triangle snap appears, then click to pin one end of the girder. (Use the Tab key to cycle through snaps until you get the middle snap.)
6. Hover and click again at B1, and then click at C1 to place the beams.
7. On the Options bar, click the Create Beam on Grid button.
8. While holding the Ctrl key down, select grids 2 and 3. The beams should appear ghosted in.
9. On the Options bar, click Finish to complete the operation.
10. In the Type Selector, click W16x26.
11. With the Beam command still enabled on the Modeling tab, click the Create Beam on Grid button on the Options bar.
12. Holding down the Ctrl key, select grids A, B, and C, then click Finish to complete the placement.

Next you will in-fill the four bays using the Beam System tool:

1. On the Design bar, click Beam System.
2. On the Options bar, select W16x26.
3. On the Options bar, select Fixed Number in the Layout box and type **2** in the box next to it.
4. On the Options bar, click Framing for the tag type.
5. With your mouse, hover over the girder that runs from A1 to A2 (you should see two dashed blue lines in the bay to the right).
6. Click the girder to finish placement in that bay.
7. Change the Layout rule to Maximum Spacing, and type **16″** in the box to the right of it.
8. Change the Tag type to System.
9. With your mouse, hover over the girder that runs from A2 to A3.
10. Click to create a placement in that bay.
11. Select Modify on the Design bar, then select grid A.
12. Click Activate Dimensions on the Options bar, then click on the temporary dimension value between grid A and grid B and change it to **45′-0″**. Notice that when the bay stretches, members are added to the bay with the Maximum Spacing layout rule while the bay with the Fixed Number layout rule simply re-spaces the two members.
13. Click Beam System again on the Design bar's Modeling tab.
14. Click the Sketch tab on the Options bar.
15. With Lines active on the Sketch menu, click the Rectangle tab on the Options bar.
16. Draw a rectangle from grid B1 to grid C2 and lock each line into place by clicking on each open lock symbol.
17. On the Sketch menu, click the Beam Direction tab, then select the girder that runs from B1 to C1.
18. On the Sketch menu, click the Structural Beam System Properties button to access the dialog box.
19. Set Layout Rule to Fixed Distance, and set the value to **8′-0″** in the Fixed Distance box.
20. Set Justification to Beginning and Beam Type to W14x30.
21. Click OK to exit the dialog box and click Finish Sketch to complete the placement.
22. In-fill the final bay with any one of the methods that you just tried.

Adding Roof Framing

Now that you have learned about how basic floor framing works, the next task is roof framing. The roof framing tools in Revit Structure are quite similar to those for floor framing. You use the Beam and the Beam System commands on the Modeling tab just as you did with floors. The big difference is that you have to slope the framing elements to fit the roof deck. As you saw in Chapter 5, you can have many varieties of straight or warped sloping decks in your projects.

So let's see how to approach and master this process by starting with a simply sloped flat roof and adding steel framing to it. First you will learn a simple approach in which framing is attached to the underside of the roof diaphragm. Then you will move on to frame more complicated roof shapes that slope from ridge lines to drain points.

Creating the Reference Level for a Sloping Roof

When you are developing your model, you create levels for your various floors. But if you have a sloping roof, where do you create the roof level, since it is not at one elevation? Best practice is to make levels at the top of wall or (for wood) at the top plate of the wall. Your ceiling framing would be there most likely as well. Another good place to create a level is at the top of roof, at the peak. That way, you project downward and see the whole roof. You will most likely need to adjust your view range so that the whole roof is showing. As it intersects at the low point, you may start to see the ceiling framing below, which makes your display confusing. In that case, you can create a filter to hide them in the view so you only see the roof members.

Attaching Sloped Framing to a Flat Plane

The first thing to do to prepare for adding the roof framing on a flat sloped roof of this type is to extend the walls and columns to the underside of the roof deck, which we discussed in Chapter 5. To attach the columns, simply highlight them in a 3D view, click Attach on the Options bar, and then select the roof deck element where you want them attached (see Figure 7.27).

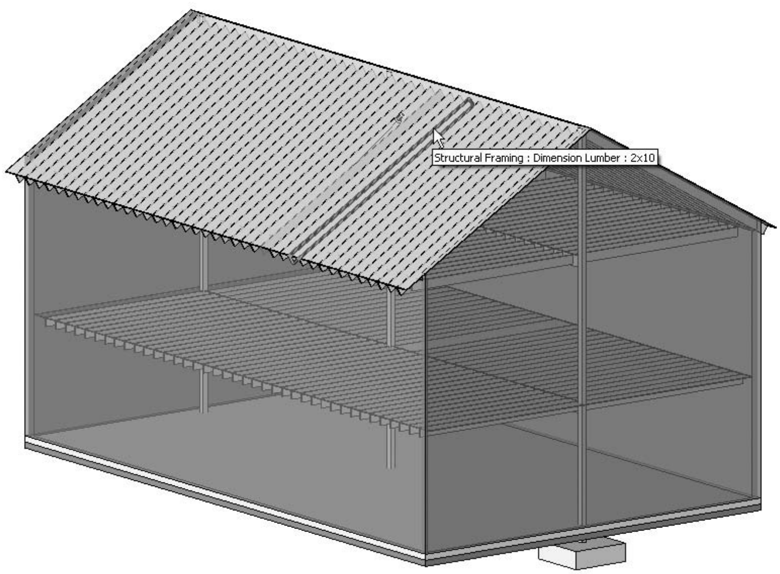


Figure 7.27
Sloping rafters added to the underside of the roof sheathing

If the roof segment is a straight sloping plane, you can attach your framing to it with this method:

1. Go to a section view of the roof deck.
2. On the left end of the toolbar, click Plane.
3. In the Plane dialog box, select the Pick a Plane option, and then click OK.
4. Click the underside of the roof deck. (You may need to activate Thin Line display to make it easier for you to see the roof surface lines in section.)
5. The Go to View dialog box will appear, prompting you to choose a plan view to display—that is, where you will draw the members in plan (see Figure 7.28). You must go directly to that view or the reference plane will not be established.

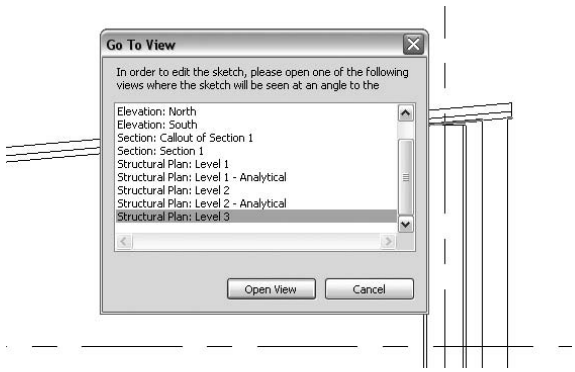


FIGURE 7.28
After selecting a plane, go directly to the view.

6. In plan view, you draw the members just as you did with the floor members, and they will attach to the underside of the deck. The advantage of this approach is that if you need to edit the slope of the roof diaphragm during the design process, increasing or decreasing it, the attached roof framing will move with it, thus avoiding a lengthy editing process.

Now you have a grasp of the basics of roof construction. The material can be wood, concrete, or steel, but the modeling principles are the same. This type of modeling will work for many of the conditions that might arise. But what about those conditions where the roof varies and is not on one plane? In such cases there is no consistent flat plane where you can attach the framing. In the next section, you will learn how to work with those types of conditions.

Warped Roof Type Framing

It is common in building design for roofs to slope from ridges to drains, creating a roof that warps and has no consistent flat plane (see Figure 7.29), as you saw in Chapter 5. Even if this is the case, there is a method of basic straight-line generation that will allow the warped surface to be framed using straight beam elements. In this section, you will learn how to model a roof like this.

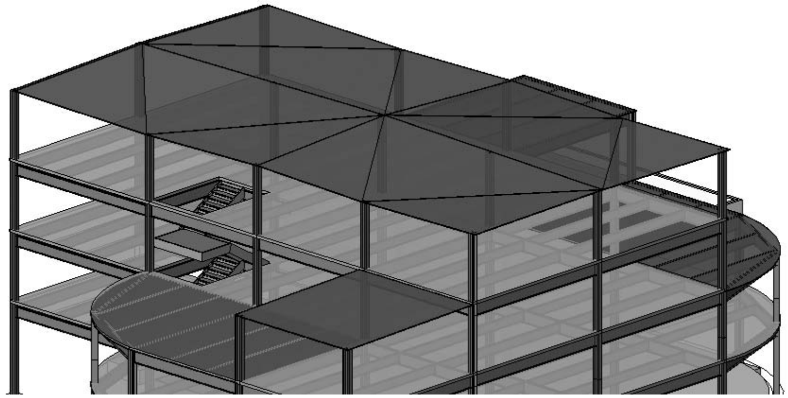


FIGURE 7.29
Steel mid-rise model with a minimally sloping roof system to accommodate drainage

This next example will use a deck that slopes about ¼″ per foot from the ridge lines. That is not a great deal of sloping, but it is enough to drain water off and is a typical type of construction for commercial and institutional-type buildings.

In Figure 7.30, you can see that the ridge lines in this example are 8″ above the roof line datum. The drain points are –4″ below the roof line datum. (The slope values are exaggerated to make them easier to see.) The rectangular area between grids 2C and 3E have ridges on three sides; then the surface slopes down on the fourth side to the drain location. When you frame that bay, the end of each beam will have a different elevation.

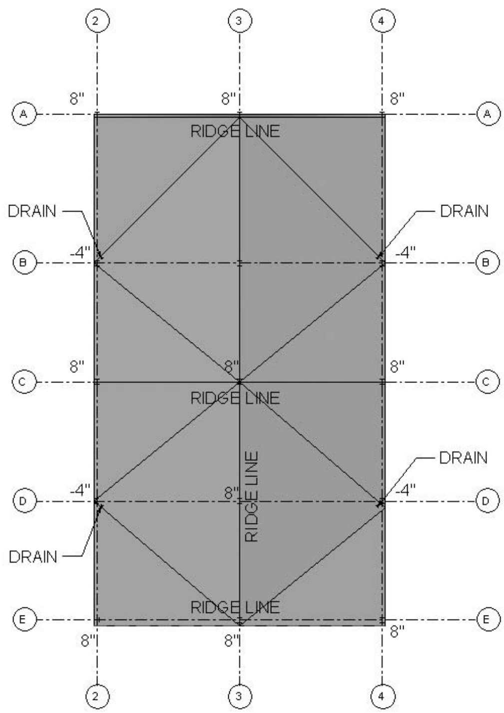


FIGURE 7.30
A roof plan sloping pattern from ridge to drain

Before you see how the bay is framed, you need to learn about 3D snapping. The next section will give you the essentials.

3D Snapping of Beam Elements

One of the most important concepts to discuss for sloping members is 3D snapping (No, that is not something you do to the guy in the next cubicle with an elastic). When you did the floor framing, you were able to set the members at one elevation, easily specified and controlled, with each end at the same elevation relative to the floor level or reference plane. But for a warped roof system with drains and ridges every beam is sloping so establishing and then editing the tops of steel can be very tedious. What tool does Revit Structure offer to help you handle this condition?

It's handled with the 3D snapping option. When you activate the Beam or Beam System command, the 3D snapping option will appear on the Options bar as a toggle box. Check it as you are creating sloping members and they will snap to the top of column, or to the intersecting sloping girder, or to a wall. That way, you do not need to compute the elevation yourself (which would take way too much time).

Once you place the beam elements, you can also edit their end elevations. When you highlight the member, the elevations at each end will be displayed (see Figure 7.31). Click the elevation and you will be able to adjust the value. That works fine when you want to edit individual conditions but you would not want to do that for an entire roof system.

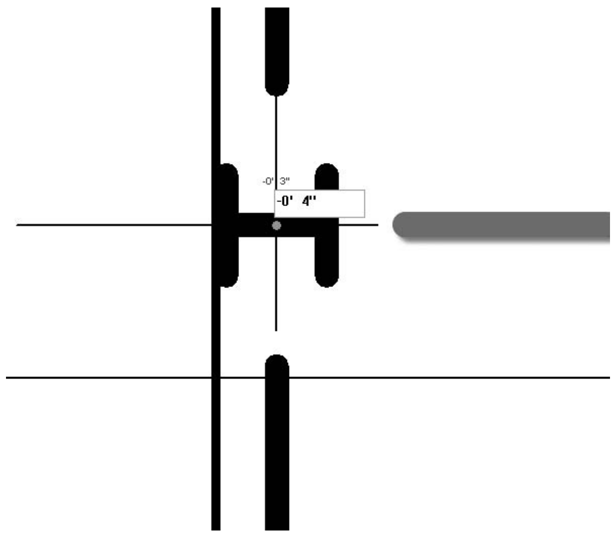


Figure 7.31
Adjusting the elevation at one end of the girder

You can also modify the value using the Element Properties dialog box by setting the Start and/or End Level Offset parameters (see Figure 7.32).

Note that 3D snapping does not work with the Grid option or with the Chain option on the Options bar.

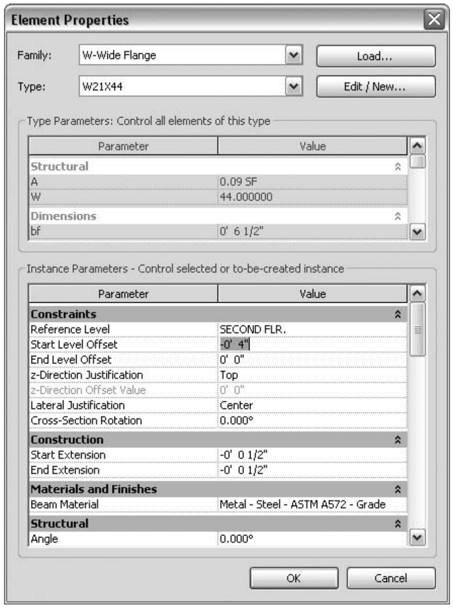


Figure 7.32
The Element Properties dialog box can be used to change beam end elevations.

Now that you understand how 3D snapping works, you can proceed to add the framing to your roof element. Here's how it is done. Since the column tops are attached to the underside of the deck, you can add the girders by snapping from column top to column top. As a result, they will be placed at the underside of the deck right where you want them without you having to compute anything. Then the framing bays are in-filled using the Beam System command, but with the 3D Snapping option checked. That way each beam will automatically snap to the elevation of the sloping girder to which it is attached, again with no need to compute any top of steel elevation. Furthermore if you edit the elevation of the sloping girder the in-fill beams will automatically flex to the new elevation.

The next exercise will lead you through the techniques step by step.

Exercise: Adding Roof Framing for a Warped Roof

First, attach the columns and walls to the roof diaphragm:

1. Open `Dataset_0702_begin`.
2. In the Project Browser, double-click the Level 3 plan view.
3. Highlight everything and then use Filter Select to gather all of the columns.
4. On the Options bar, click Attach, and then select the roof diaphragm.
   The columns will stay attached unless you detach them, even if the deck shape changes. Of course, now each column top is at a different elevation on the sloping surface, which is difficult to calculate manually.

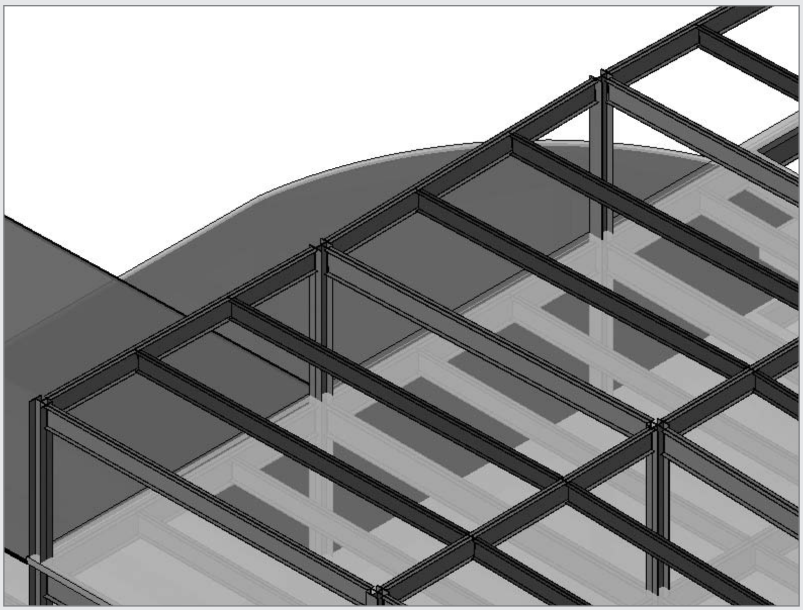
Next, add girders between the columns:

1. On the menu bar, click Settings ➢ Snap.
2. In the Snap dialog box, uncheck all but the Points snap, and then click OK to exit.
3. Click the Beam command on the Modeling tab, and then select a beam size from the Type Selector.
4. On the Options bar, check the 3D Snapping box.
5. Hover over a column top to activate the Points snap.
6. Click to pin one end of the beam; then click the other end at another column.
7. Keep adding girders until all the columns are connected. The girders are all sloping from column top to column top.

Next, you will in-fill the bays with sloping beams:

1. On the Modeling tab, select the Beam System command.
2. On the Options bar, check the 3D Snapping box.
3. On the Options bar, set an 8´-0˝ Maximum Spacing as a layout rule.
4. Set the direction of the framing.

When you finish, your framing should look like the following illustration in a 3D view:

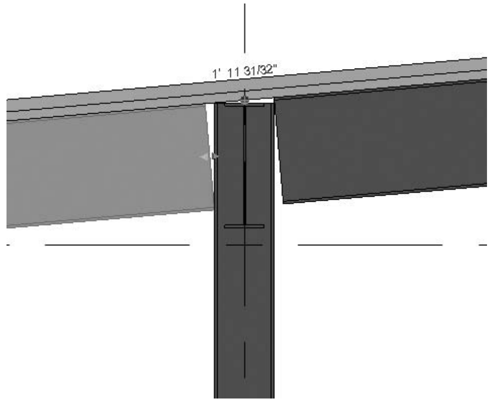


When you want to make larger-scale sections of sloping members, you will find that the ends of the beams needs to be adjusted in order for them to show correctly in your view, as the next section will demonstrate.

Shaping the Ends of the Sloping Members

Look at Figure 7.33 and notice that the ends of the sloping girders are not cut vertically as you would need for connecting the shear plate to the column. It is an important consideration for you when you are trying to detail the connection in a section.

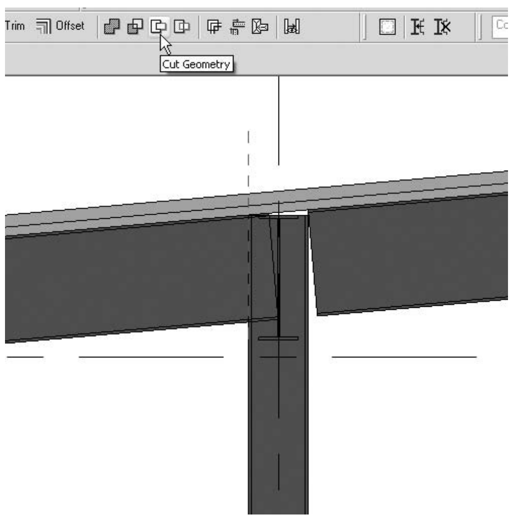
Figure 7.33
The girder is not cut correctly at the column.



To correct the display, you have to create a reference plane at the edge of the column flange to cut the beam vertically:

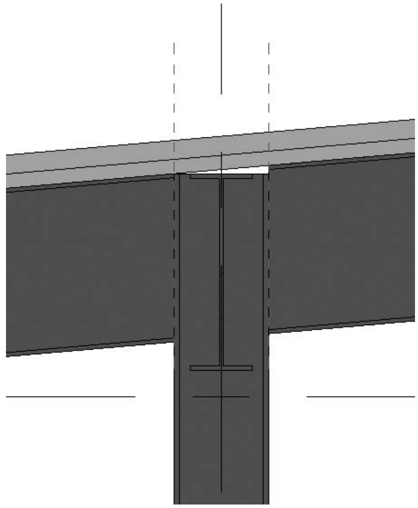
1. Highlight the girder, click on the blue triangular shape handle, then pull the end of the girder over the column flange. (This will not affect the end node location of the girder.)
2. Draw a reference plane on the outer edge of each column.
3. Click the Cut Geometry tool on the toolbar and then select the girder and reference plane (see Figure 7.34). That will do it.

Figure 7.34
Extend the beam and add a reference plane.



The final display will look like Figure 7.35. Also notice that the column and girder do not fully intersect with the roof. This is fine; it's how it would be constructed in reality. The framing members should be vertical and will have bent plates attached to the top flange to connect to the sloping surface of the roof deck.

FIGURE 7.35
The final section with shaped girder ends



Adding Steel Braced and Moment Frames

Now that you have added your floor and roof framing, you will start work on your moment or braced frames depending on which system your project is using. These frames will accommodate the seismic system you might be using on a steel or concrete building, depending on where your building will be located in terms of seismic zones.

Moment Frames and Cantilevered Beams

When you model moment frames, you use standard framing techniques as described earlier in this chapter, but the display must be adjusted to add Moment Connection End symbols on the ends of the frame members. You might also want to display the moment frame girders with a heavier line weight, which you can do by assigning them to a particular structural category or through use of a Filter.

One way is to assign the moment frame members to the Other category for that use, as was described earlier. Another way is to build a Filter. Filters are very powerful. After you place the frame member put an entry such as "MF" in the Comments parameter. That will identify that instance as a frame member for which you can then create a Filter. You define the Filter so as to display the member with a heavy lineweight.

Though not a moment frame item, cantilevered beams that are moment connected need a similar symbol, an open triangle, added to one end of the element. The symbol works just like the moment frame symbol. The annoying part is knowing which end to select in the Element Properties dialog box. The parameter is either for the start or the end that you placed the member, but who remembers that when you come back to add the symbol? It is best add the symbol right after placing the member, otherwise it becomes guesswork.

To add moment frame or cantilever moment symbols, highlight the member and click Element Properties on the Options bar. In the Instance Parameter area of the resulting dialog box, you set the Moment Connection Start and/or End value to add the symbol to your project (see Figure 7.36).

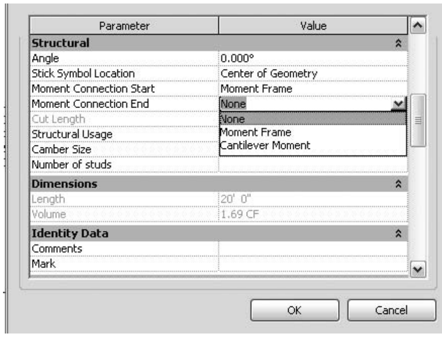


FIGURE 7.36
Adding the Moment or Cantilever symbol to an element

One factor you should consider when displaying the symbols is the distance of the symbol from the column in your course mode plan view. If it appears too far from the column or overlaps the column, you can adjust the distance. Choose Settings ➢ Structural Settings from the menu bar and change the Symbolic Cutback distance for Beam/Truss to a value that displays well on your plan view. Be careful, though, because this will change the end distances for all your beams in the project throughout all views.

The Structural Settings dialog box helps control the display of brace and moment frame symbols (see Figure 7.37). If the symbols do not correspond to your company standards, you can change the default version of the symbol or make your own. You enable your own symbol in the Structural Settings dialog box for the entire project.

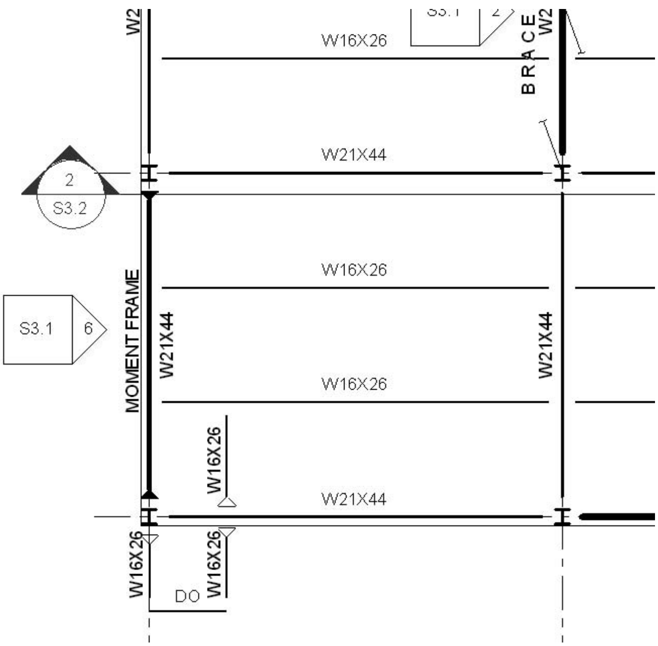
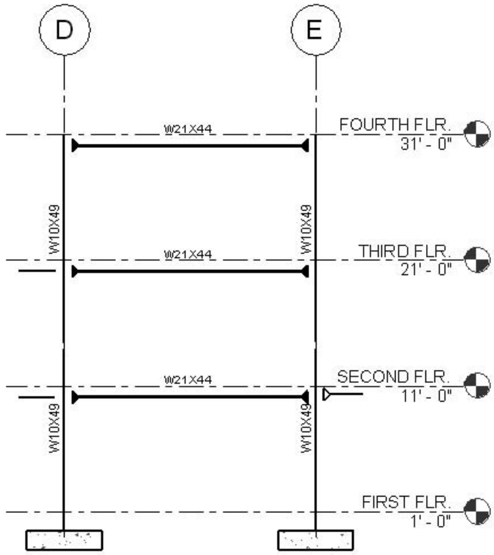


FIGURE 7.37
Moment frame and cantilevered steel framing displayed in plan view

Another common display problem is that the cantilever open triangle appears too thin in plan view, and it can be quite a frustrating adventure trying to find out where to adjust its line weight. To adjust the open triangle line weight, click Settings ➢ Object Styles ➢ Annotation Objects ➢ Connection Symbols and adjust the projected line weight value. This should also be done in your company template so that you standardize the display. For more information on standards, see Chapter 17. Figure 7.38 shows how the display should look when you finish.

FIGURE 7.38
Moment frame elevation in coarse mode



Projects using steel moment and braced frames usually require creation of frame elevations. This is an important consideration on any project, and Revit Structure has the special Frame Elevation option to make the process easier. The Frame Elevation command can be found both on the Basics and View tabs. It creates an elevation of the frame and is attached to a gridline. Figure 7.38 shows a typical moment frame displayed in coarse detail mode where the frame members display as stick symbols. The floor and roof categories have been shut off in this view so that only the frame shows but the foundations remain. You could also display the moment frame in medium mode with the steel members showing their real shape and size, and floors and roofs turned on for a realistic view. But medium mode might not be the best option because you don't see the moment connection symbols as you do in course mode and so the frame members are not as easily identified.

Those are some of the issues with creating moment frames and cantilevered framing in your virtual building. Next, you will examine the process for creating braced frames, which involves much more elaborate modeling techniques.

Contending With Many Types of Framing

You may have been to some of those sales presentations where framing is quickly and easily added, and where every condition is covered, and everybody is really happy. Well, in real projects it just ain't so! A typical structural project will most likely have all sorts of odd framing conditions that fit no pattern and many types of different framing to place. You will find that the 80-20 rule governs. Eighty percent of the framing goes in quickly and easily, but the other 20 percent is a mind bending frustration to create. And your project fee will be spent in a corresponding fashion. As an example let's consider this real project.

This recent project is a new high school campus in California. The campus has two large multilevel classrooms. The classroom floors are composite metal deck and concrete, with steel framing members and wide flange steel columns. The seismic system consists of steel-braced frames. The roofs were sloping at ¼″ per foot and were constructed of 1½″ metal roof deck and lightweight concrete fill. Tube framing jutted out from the roof to support large sloping parapets. The corridors in the floors were depressed, as was the framing.

The gymnasium structure has large steel trusses supporting the roof. The library/administration building is a two-story braced frame structure using HSS tube columns. A long canopy snakes through the campus. A subterranean parking garage is constructed with a precast concrete double-T roof that has basketball courts on top. A one-story cafeteria structure with very complicated canopies completes the campus buildings.

You will encounter numerous types of framing on your projects, and it can get quite complicated. You need lots of tools in your Revit tool chest to get it done, as well as a good deal of imagination. But after you get familiar with Revit Structure you will find that you can accomplish most any condition you encounter.

Braced Frames

Once the floor and roof framing is completed, you are ready to add vertical braced frames (see Figure 7.39) to your model. . If you are using the analytical mode you will also want to set the Structural Usage for the floor members of the frame to Horizontal Bracing.

Braced frames need to be added in an elevation view. Usually that will be a Frame Elevation as discussed earlier. The most important point to understand is that the diagonal members need to be connected to Work Points. The work point is usually at the center of the member that it intersects, such as a girder. As you place the diagonal member you will see Revit trying to snap to those points. Pay close attention because lots of times it will try to snap to the adjacent member so you need to be careful.

The following exercise will lead you through the construction of a braced frame so you can see how the process works.

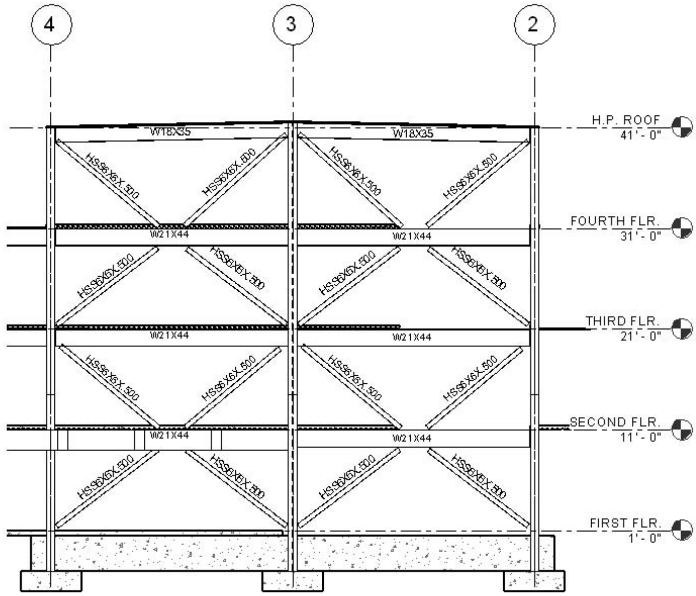


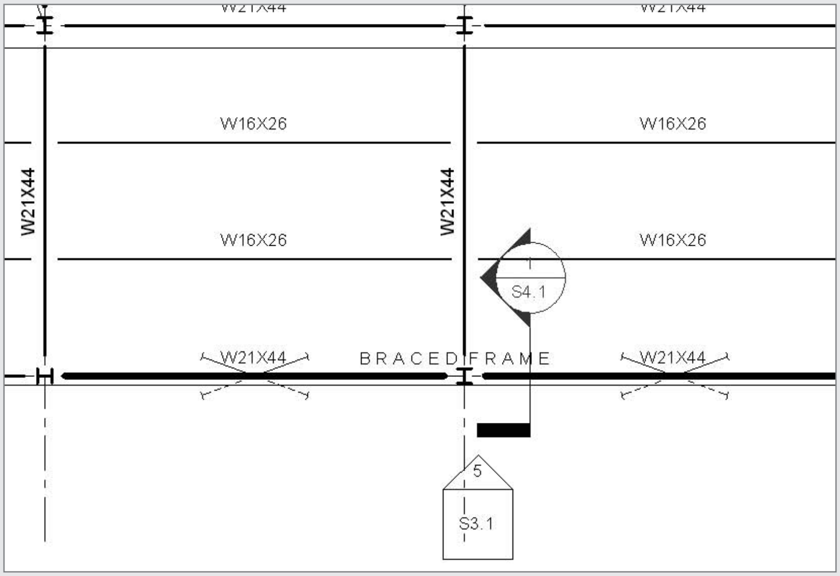
FIGURE 7.39 Steel x-braced frame elevation with no connection plates shown

EXERCISE: CREATING A STEEL-BRACED FRAME

In this exercise, you will add an x-braced frame to a project.

1. Open `Dataset_0703_begin`. Level 1 plan view should be active.
2. Click Framing Elevation on the View or Basics tab.
3. If needed, use the Type Selector and choose Building Elevation.
4. Hover over the gridline and click to select the direction of the elevation so it is facing upward on the screen.
5. Go to that elevation; then on the View Control bar, change the detail mode to medium.
6. Highlight the crop box and use the shape handles to pull it past the ends of the frame.
7. On the Modeling tab, click Brace.
8. In the Type Selector, select an HSSS6X6X.500 as the diagonal brace member.
9. Hover over the base of a column and click on the middle bottom of the column (if snaps are turned off from the previous exercise please turn them all back on).

10. Hover over the middle of the second floor beam until you can snap to the work point, as shown here:



11. Repeat the sequence until the x-frame is completed.
12. Where the ends of the diagonal braces overlap the floor beams, highlight each one; then click and drag the triangle to move the end of the brace until it clears the beam.
13. In the Project Browser, activate Level 3 plan view.
14. On the menu bar click Settings ➢ Structural Settings.
15. Change the Brace Symbol plan representation to Line with Angle, and then exit.

The brace symbol will automatically be placed in plan view after you place the brace in elevation. You can use two types of plan symbol for braced frames: Line with Angle or Parallel Line. The Line with Angle option shows more information about the braced frame because it shows the start point of the brace and the direction it is going (see Figure 7.40). It can be important to know how the frame members are located in case there are door or window interferences on the architectural side of the design. Line with Angle allows the architect to easily see the directions without referring to the frame elevation. A solid brace symbol indicates it is going up, while a dashed symbol indicates the brace is going down. Unfortunately, Revit Structure does not ship with a dashed line symbol for braces going downward, but you can easily create one by editing the default brace symbol and saving it with a new name.

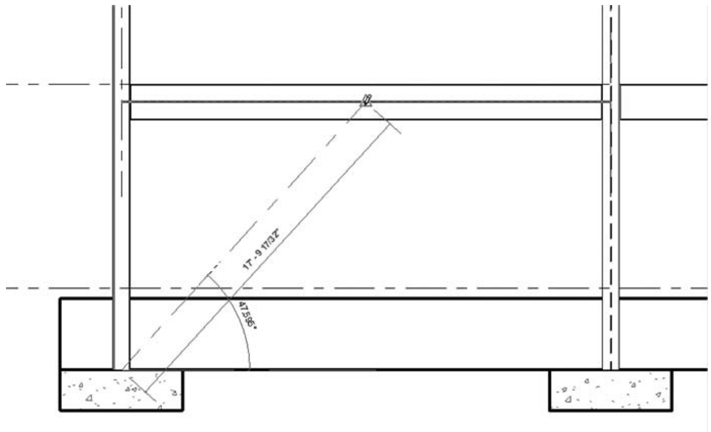


FIGURE 7.40
A braced frame with Line with Angle symbols showing braces up and down in plan view

You might also wonder about the braced connection gusset plates. They are not shown unless you add them. It has to be a judgement you make for your firm as to whether you show them in your elevation. Whereas it was best practice to show the moment frame in course mode in order to see the moment connection symbols it is best practice to show the braced frames in medium mode. That is because braced frames are much more likely to interfere with the architecture. The frame gusset plates can sometimes get very big and also interfere with adjacent windows or doors so in those cases it might be a good idea to model them.

That ends our discussion about Framing in Revit Structure. You have gotten a thorough overview of the capabilities and the frustrations that you will encounter along the way as you design your project. But don't be intimidated. Revit Structure is a great program and quite intuitive. If you keep adding tools to your Revit tool chest, before you know it you will be a master.

The Bottom Line

Understand structural framing families and properties. Revit Structure modeling is a constraints-based system that allows the model to update as changes occur, keeping the overall relationships between elements the same.

Master It Describe the two basic modeling constraints for attaching beams and braces and why they are important.

Add floor framing. When you add floor framing to your project, you probably start with a fuzzy idea of the size and initially use a placeholder. As the design progresses and comes into sharper detail, you will update the sizes and spacing in many cases. The model must have a maximum of flexibility to make the editing practical.

Master It You are in schematics and know the bay widths on your building will change considerably. You want your framing members in each bay to be about 10´ from center to center no matter how wide the bay becomes during the course of the design. What layout rule is the best to use in this case?

Add roof framing. Roof framing must support roofs that slope from ridges to drains. That means all the support beams and girders must slope as well. During the design process the roof can change in shape and slope. Costly editing can eat away your at your design fee.

> **Master It** Calculating the end elevation for each sloping beam would be too time consuming and a nightmare to edit. What process do you use to most efficiently place the roof support system?

Create moment and braced frames. Moment and braced frames are an important element of many structural designs. Revit Structure has two methods of displaying the braces in a plan views: Parallel Line and Line with Angle. The symbols are placed automatically in plan view as you draw the braces in elevation views.

> **Master It** Which display type is the most informative of the braced frame layouts and how do you set it to display correctly in the plan view?

Chapter 8

Foundations

This chapter focuses on the creation of foundations in Revit Structure. The topic of foundations is quite a broad subject, and covers any element associated with the support of a structure. Items discussed in this chapter include footings (spread and strip), foundation-bearing walls, foundation-retaining walls, structural slabs, and large operational recesses such as elevator pits and mechanic bays. Also included are items such as grade beams, piers, pilasters, caissons, and pile systems.

It is important to note at the onset of this chapter that reinforcing will not be added to the foundation. Reinforcing is such a large topic it occupies an entire chapter (Chapter 10).

A key take-away in this chapter is drawing aesthetics. As a firm moves to Revit Structure from a CAD-based platform, one of the defining questions of a successful migration is "Do my drawings in Revit look the way they did in CAD?" Many firms base their success with Revit almost entirely on this requirement. Firms take years to develop typical details, notes, symbols, and schedules that need to be incorporated into Revit Structure. This chapter will also focus on incorporating such details.

To begin, it is appropriate to start with, well, walls and columns. It may seem redundant. After all, we have covered both walls and columns in earlier chapters, but foundation walls are certainly different than the architectural walls, masonry-bearing walls, and shear walls we looked at in previous chapters. Also, why columns? Well, concrete columns, when placed in a foundation, can become piers and pilasters with a simple modification to the default family that Revit Structure offers.

So, this introduction has rattled off several topics, all of which are contributing factors to a foundation. Let's get started!

In this chapter you will learn to:

- Create strip footings
- Create footing steps
- Create and add foundation slabs
- Create grade beams
- Create elevator pits

Foundation Walls

From the start it is important to point out that foundation walls, when placed in the model, start from the top and go down. This can be off-putting at first, but the nature of a foundation

concrete-bearing wall is that you are standing on the top of the wall looking down. This is a good thought process as we go through the exercise, because Visibility parameters play an important role in the visual aspect of the model.

To begin, open a new model or find a model you have been working on. You will need to add a level down from the Level 1 elevation. This will be your Top of Footing level. If you are thinking "Geez, this should be set up in a template already," you are well on your way to being successful at Revit Structure. (See Chapter 2 for more on template setup.)

While placing this foundation wall, if you are in Level 1 and you intend on placing this wall from Level 1 to the Top of Footing level, you are correct. Always think down. In the Basics tab on the Design bar, click the Structural Wall button. This of course starts the Wall command.

To make this a foundation wall, select Basic Wall: Foundation – 12" Concrete. Once you choose the foundation wall from the Type Selector, the Options bar changes slightly. You can no longer choose between the depth and the height; only the Depth setting is enabled. For Depth on the Options bar, select the T.O. Footing level. This will allow you to place the wall the way you want. If you have been experimenting with placing a foundation wall, you may have noticed that in the element properties of a foundation wall the Top Constraint and the Base Constraint values were the same. The Base Constraint, however, has an offset. This is the default for the placement of a foundation wall. We recommend that you use levels as wall constraints and only rely on offsets if unavoidable.

To place a foundation wall, follow these steps:

1. Create a T.O. Footing level if you have not already done so.
2. Change Display to the level that represents the top of the wall.
3. In the Basics tab on the Design bar, select the Structural Wall command.
4. On the Options bar, select Basic Wall: Foundation – 12" Concrete from the Type Selector.
5. Again on the Options bar, select the level below (T.O. Footing) for the base constraint.
6. Create your walls.

Once you have set the walls and specified the constraints, you will find it much easier to place piers and pilasters along with other items that are dependent on the wall height and parameters.

Piers and Pilasters

Placing piers and pilasters is much like placing concrete columns. That makes sense because you *are* placing concrete columns. The reason you should apply this method is twofold. One, the analytical model will be in place for whatever analysis application you intend on sending the model to. Two, you can constrain the tops and the bottoms to the levels. This kind of functionality is exactly why we are using Revit Structure in the first place. Since piers and pilasters are sometimes not perfectly square, Revit Structure provides various geometric shapes. To find a suitable shape from the default Revit Structure library, choose File ➢ Load from Library ➢ Load Family, as shown in Figure 8.1. From there you can browse to the default location where you are keeping your library.

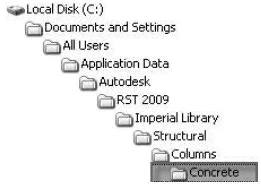


FIGURE 8.1
The folder structure

In the library, you can browse for the column shape you want. If you don't see the exact shape you want, you can customize these families to fit your application. To follow along with our example, select `Concrete-Square-Column.rfa`. Because this column has the same material as the wall it is being placed into, the concrete will clean up itself and join the wall and the column together upon placement, as shown in Figure 8.2. Once the family is loaded into the model, select the Concrete Square-Column: 24 × 24 type from the Type Selector on the Options bar. On the Options bar, notice you can change the Height setting to Depth. Change the Depth setting to T.O. Footing. Simply place the column on the wall. Congratulations! Your column is now a beautiful pilaster.

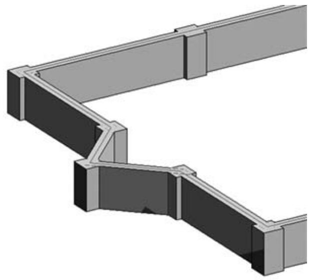


FIGURE 8.2
Pilasters added to the foundation

Placing a pier is just as easy as placing a pilaster. Normally, the pier geometry is similar to the pilaster geometry, so the same column family tends to play a dual role. Also, more than likely, you are going to have a column grid set up. In the plan (of course we are still thinking "top down"), drop a column in the same manner as the wall pilaster into your grid. If you do not have a grid, you can always place the pier, and then move it around after the placement. From there, you can use the conventional Copy command to arrange the piers as you need them.

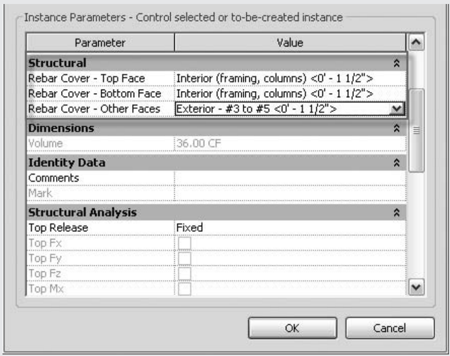
The steps for adding a pilaster or pier are as follows:

1. If the desired column is not loaded into your model, select File ➢ Load from Library ➢ Load Family, and then browse to the Structural folder in the Imperial Library directory. From there, navigate to the Columns folder and then to the Concrete folder.
2. Once the pier is loaded, click the Structural Column button on the Basics tab on the Design bar.
3. On the Options bar, select the appropriate column from the Type Selector.
4. Place the column into the foundation wall. If it is a pier, place it on a structural grid, or place it manually in the model.

Real World Scenario

Reinforcing Considerations

Remember, items such as piers and pilasters are not automatically reinforced. They will be, but reinforcing is a separate procedure. It will be covered in depth in Chapter 10. For now, however, we can include some provisions to make the process of reinforcing these items much easier and more accurate as well. Any concrete element in Revit Structure has built-in *reinforcing cover settings*. That means that the minimum allowance for reinforcing to be set back from the faces, the tops, and the bottoms of walls and foundations can be applied to the wall and foundations themselves. The typical defaults are 1-½″ for the faces, tops, and bottoms of bearing walls and 3″ for an earth-formed item. You can adjust these settings on an element-by-element basis. To examine the reinforcing cover settings, select the item, such as a pier, a pilaster, or a foundation wall. Click the Element Properties button, and scroll down in the resulting dialog box until you see the Structural category. Here you can view the cover settings as they relate to this instance. You can change them to the cover settings preconfigured in the template.



Footings

Now we've come to the fun stuff: footings. The two most popular types of footings will be covered early on: Strip (continuous) and Spread (isolated). Each behaves in a similar fashion, but you approach them differently.

Strip Footings

There are two different kinds of strip (continuous) footings. One is a bearing footing; the other is a retaining footing. Each has separate properties and placement methods and will be discussed separately. The greatest advantage to using Revit Structure for footing placement is the fact that a footing "knows" it's a footing. It knows that when you place it under a wall, it belongs with that wall regardless of the fact that the wall may move, change shape, or change type. A strip footing knows to do this, because a relationship is established between the strip footing and the wall it is placed under. Also, when the wall is deleted, so is the footing. A strip footing will not

exist unless there is a wall bearing on it. This is why, when you choose the command from the Modelling tab, it actually says Foundation ➢ Wall. It may be misleading at first, but the understanding is that this is a footing that bears the weight of a wall.

As a footing is placed in a model, it is good to have the View properties preset to allow for the viewing of an item that may be below the actual level you are in. For example, if you are in a basement level, and the footing you are placing is on the Top of Footing level, Revit Structure may report an error saying that the items placed are not visible in this level. It goes on to suggest that you turn the items on, but the real culprit is the fact that your View Range option is set too shallow, as shown in Figure 8.3. By pressing VP (for View Properties) and choosing the View Range command, you can use the resulting dialog box to tell Revit Structure to either look to the floor below or look down past that floor to an unlimited range.

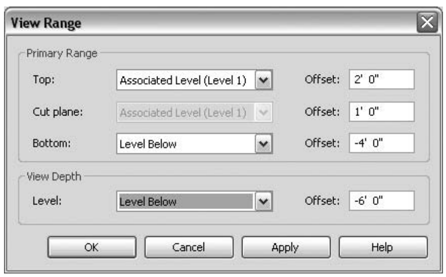
The steps to take when beginning to place a wall foundation are as follows:

1. You need to have walls in the model. They don't have to be complete, or even constrained to levels (although we recommend that you do).

Be sure the View Range option is set so you can see the footing below.

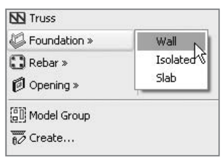
2. Press VP, or right-click in the view window.
3. Scroll down to the View Range option in the Extents category.
4. Click the Edit button. In the View Range dialog box, in the Primary Range category, set Bottom to Level Below.
5. The View Depth level should also be set to Level Below. If your footing is deeper than the level below, it may be a good idea to select Unlimited for both the view Depth and the Level Below.

FIGURE 8.3
View Range in the View properties



6. Go to the Modelling tab on the Design bar and select Foundation ➢ Wall, as shown in Figure 8.4.

FIGURE 8.4
The Modelling tab is used to launch the footing command.



7. On the Options bar, select the type of footing you want. By default you will have two choices: a bearing footing (centered under the wall) or a retaining footing, where you can specify a toe and a heel length.
8. Select the wall(s) you want the footings to attach to.
9. As you are placing the footings, it is important to reiterate that the Foundation command is running until you tell Revit Structure you are done. Revit Structure shows this to you by keeping the Foundation button engaged by displaying Wall.

Easy enough, right? Another nice feature afforded to the Foundation command is that Revit Structure understands multiple placement of strip footings. For example, if you have more than one wall (and I hope you have more than one wall!), you can start the Foundation ➤ Wall command. Hover over a wall but do not click it. Once the wall becomes highlighted, press the Tab key on your keyboard. This will highlight every connected wall, as shown in Figure 8.5. Your cursor tooltip will read “Chain of Walls or Lines.”

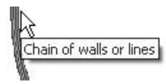


FIGURE 8.5
Tab to select multiple walls.

Once all of the walls are highlighted, you can then click. As shown in Figure 8.6, a footing will be placed under all the walls. If the wall is stepped (see Chapter 7 for more on stepping a wall system), the footing will accommodate the step. Later in this chapter we will discuss adding a stepped footing family both as an inserted foundation and as an in-place footing step.

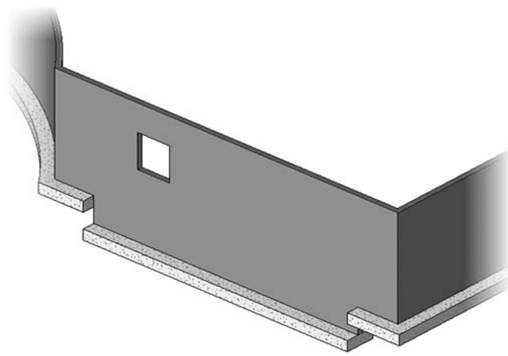


FIGURE 8.6
Wall footing

Another aspect to the functionality of placing a strip footing is, of course, the Options bar. As you are placing the footing, check the Multiple option. Then, you can select the entire model with a left-to-right crossing window. After all the walls have been selected, click the Finish button. A footing is automatically added to the bottom of the selected walls. Remember that after the walls are selected, and you have clicked Finish, Revit Structure is still in Foundation mode. Either press the Esc key on your keyboard, or choose the Modify command from the Design bar.

The next section will focus on the difference between a bearing footing and a retaining footing. Both are categorized as wall foundations.

Bearing Footings

It's time to examine the properties of a bearing footing. A bearing footing is defined by a single width, a foundation thickness, and a default end extension length. When a bearing footing is placed under a wall, it will be centered by default. To offset a bearing footing, you must specify an Eccentricity increment in the instance parameters of the element properties. There are two ways to gain access to the element properties. One is to issue the Foundation ➤ Wall command from the Modelling tab on the Design bar. Once the Foundation Wall command is running, on the Options bar, click the Element Properties button. Once in the Element Properties dialog box, click the Edit/New button. At this point, we recommend that you click the Duplicate button and rename the footing. This will help prevent much confusion among the design team members. Once you have defined a new footing, you are free to make changes to its type properties. For instance, if a 36″ bearing footing is too big, and you wish to create a 24″-wide footing, follow these steps:

1. On the Modelling tab of the Design bar, click the Foundation button. From the flyout menu that appears, select Wall (see Figure 8.2).
2. With the Wall command running, make sure that Bearing Footing appears in the Type Selector on the Options bar.
3. Click the Element Properties button on the Options bar.
4. Click the Edit/New button in the Element Properties dialog box, then click Duplicate.
5. In the Name box, type **Bearing Footing - 24″ x 12″**, as shown in Figure 8.7, and click OK.

Figure 8.7 Naming your footing

6. In the Type Properties dialog box, as shown in Figure 8.8, change the Width setting to 24″. Once you do this, a new footing is born. Click OK twice to leave the Element Properties dialog box and return to the model. You can now place the footing.

Another way (and the recommended way) to add and configure a new footing is to access the foundations in the Project Browser, as shown in Figure 8.9. This way allows you to better understand exactly how many footings of this type are in the model, and it helps prevent an unintentional duplication or redundancy. It also "forces" you to stroll through the Project Browser, giving you a quick glance at other families and systems that could help…or that are causing issues. Simply expand the Families tree in the Project Browser and navigate to the Structural Foundations category. You will see the two bearing footings listed as well as a retaining footing. If you know you want a duplicate of a footing, simply select the footing, right-click, and select Duplicate from the context menu.

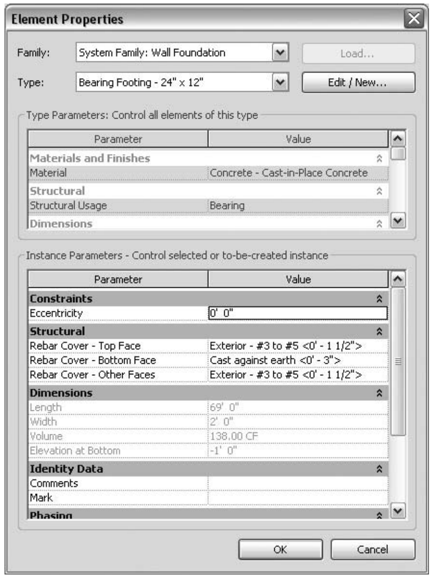


FIGURE 8.8
Element properties for the bearing footing

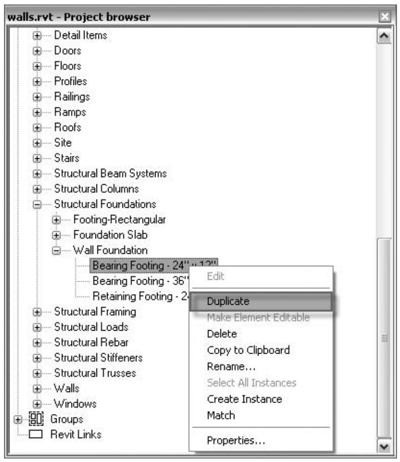


FIGURE 8.9
Structural foundations in the Project Browser

Once you duplicate the foundation, Rivet Structure places a copy in the Project Browser. Double-click on the copy, and then modify the type parameters.

Another advantage to this method is that you can place a footing by simply dragging a foundation family into the drawing area from the Project Browser. This allows you to bypass the Design bar altogether. Also, all of the cool kids are doing it this way.

To step through the process:

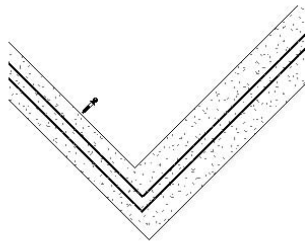
1. Navigate in the Project Browser to the Structural Foundations category.

2. Drill into the Structural Foundations category until you see either the footing you want to place in the model or a similar footing you want to copy.

3. Right-click the footing.
4. Choose Duplicate from the context menu.
5. Double-click and rename the new footing, and click OK to close. Then use the type properties to modify the footing.

Another advantage of controlling your model from the Project Browser is you can actually right-click on a footing in the Browser and select Match, as shown in Figure 8.10. Revit Structure turns your cursor into a dropper icon. Find a footing you want to apply the properties of the footing family to, and Revit Structure will change the footing you select. Of course, always look at the Options bar. You can select the Multiple option and use a select window.

FIGURE 8.10
Matching a footing to the family



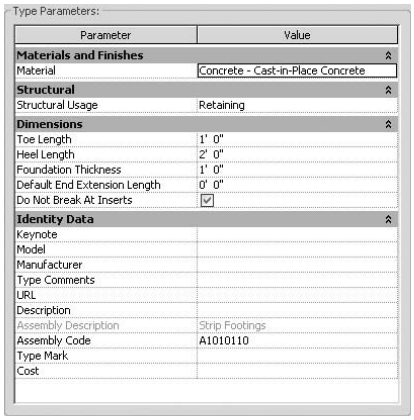
Not every footing in your model will have a uniform load bearing upon it. There are, of course, situations where lateral pressures influence the design of the footings. In Revit Structure, you can choose to use a retaining footing in this situation.

RETAINING FOOTINGS

Retaining footings have additional properties that bearing footings do not. One of the most common mistakes while placing a bearing footing is that a retaining footing is sometimes mistakenly inserted instead. The two types are similar, but will obviously affect the model in different ways. Inserting a retaining footing is exactly the same as inserting a bearing footing in that you can either add it from the Modelling tab of the Design bar or you can find it in the Project Browser. The only real difference between the two types of strip footings is the fact that, in a retaining footing, you can specify a toe length and a heel length, as shown in Figure 8.11. These lengths are based on the center of the wall that the footing is under. There is no total length, so you have to be careful in terms of the name. The default name for a retaining footing reflects the total width. You do not have to follow this naming convention.

To insert a retaining footing, follow these steps:

1. Click the Modelling tab on the Design bar, and select Foundation ➢ Wall.
2. On the Options bar, select Retaining Footing - 24″ × 12″ × 12″. Note that you can, and should, create a new retaining footing. Defaults are a good start, but do not rely on them to complete your design.
3. Select a wall to place the footing.



Type Parameters:

Parameter	Value
Materials and Finishes	
Material	Concrete - Cast-in-Place Concrete
Structural	
Structural Usage	Retaining
Dimensions	
Toe Length	1' 0"
Heel Length	2' 0"
Foundation Thickness	1' 0"
Default End Extension Length	0' 0"
Do Not Break At Inserts	☑
Identity Data	
Keynote	
Model	
Manufacturer	
Type Comments	
URL	
Description	
Assembly Description	Strip Footings
Assembly Code	A1010110
Type Mark	
Cost	

FIGURE 8.11
Type Parameters for a retaining footing

In many cases, our site is going to slope in some manner, causing the minimum footing depth to be compromised. In this case, of course, you need to step the wall and the bearing footing as well to maintain a consistent frost line cover.

STEPPED FOOTINGS

A continuous footing, as discussed earlier, will follow the contours of a wall. This behavior is crucial if you're using a stepped footing. In such a case, we recommend that you follow the procedure for stepping the wall in a straight-on elevation. Once in the elevation, select the wall to be stepped. Click the Edit Profile button on the Options bar, and you can then sketch the steps by drawing lines in the Sketch mode of the wall. (See Chapter 6 for more about this process.) The nice thing about this process is that if you place a footing underneath the wall before sketching in the steps, the footing will automatically adapt to the new profile.

The steps are as follows (pun intended):

1. Find the elevation in which the wall is to be stepped.
2. Select the wall and click the Edit Profile button on the Options bar.
3. Sketch the steps similar to Figure 8.12. Once you have finishing sketching the profile of the wall, click Finish Sketch. Your wall will be similar to Figure 8.13.

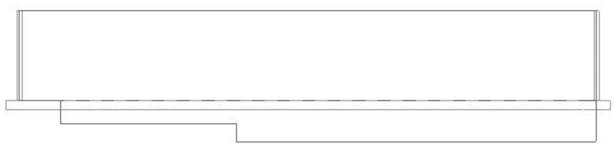


FIGURE 8.12
The wall during Sketch mode (with the footing in place)

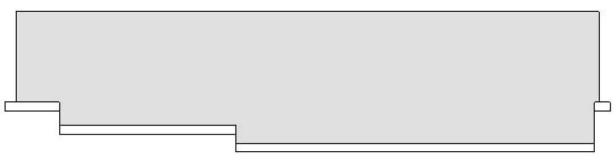


FIGURE 8.13
After finishing the sketch

Notice the footing is along for the ride. One glaring issue, however, is the angled footing step. A backhoe is not going to dig a ledge like the one in Figure 8.13. There is going to be an angle extending down from the bulkhead, angling into the footing at the lower elevation. There is something we can do, though: it involves using a family. There are two ways you can proceed. To add a stepped footing family, you have to create it. We have provided one for you at the book's companion web page, `www.sybex.com/go/masteringrevitstructure2009`. Locate the family file (`.rfa`) called `footing step.rfa` and copy it to your Revit Structure library under `Documents and Settings\All Users\Application Data\RST 2009\Imperial Library\Structural\Foundations`

1. In Revit Structure, select File ➤ Load from Library ➤ Load Family, as shown in Figure 8.14. You can then find the family in your default Revit Structure library. If you are at work, this directory may be (should be) on the network.

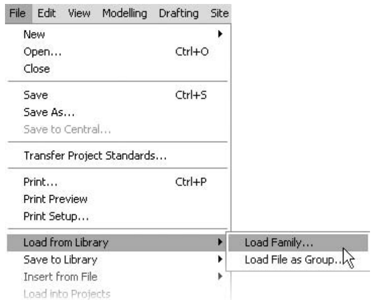
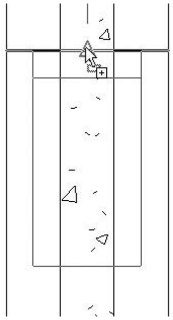


FIGURE 8.14
Selecting File ➤ Load from Library ➤ Load Family

This family has been provided for this chapter. If you would like to learn how to create a stepped footing, see Chapter 18.

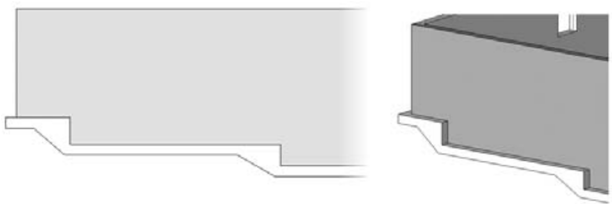
2. Once the family has been loaded into the model, you need to be in a plan view in order to insert it. The beauty of stepping a footing in Revit is the fact that, in plan, the lines show up for the step. The elevation and the plans are always fully coordinated.
3. Using this edge as a guide, click the Modelling tab on the Design bar.
4. Click the Foundation button, and then select Isolated from the flyout menu. The Type Selector in the Options bar should display Footing Step. As you are placing the footing step in the model, you may have to press the Tab key to position the step family in the orientation you need, as shown in Figure 8.15.
5. There will be a 6″ offset line indicating where the bottom of the footing should align. Find the midpoint of the footing that is in place, and click the point.
6. Once the footing is snapped in place, you must adjust the width. Select the footing family, open its Element Properties dialog box, click Edit/New, and change the width to the correct footing width. In Figure 8.15, the footing step is 2′-0″ wide. The foundation is 3′-0″ wide.
7. In the properties of the footing, change the width to 3′-0″.

FIGURE 8.15
Dropping in the step footing



8. Now the height needs to be adjusted. Go to the elevation that displays the step. In the elevation, determine what the step height needs to be. (Chances are, the footing will not be aligned with the rest of the foundation; we will move the step down in a moment.)
9. Select the footing step and open its Element Properties dialog box. Click Edit/New, and change the height to the same increment as the drop in the wall.
10. Move the stepped footing down to align with the bottom of the lower step. The concrete will join itself, as shown in Figure 8.16. You can now copy the step to any other location.

FIGURE 8.16
Completed step



Because we were able to insert this footing as an isolated footing, the next step in creating a foundation will be similar. As you are placing footings in the next segment, remember that isolated footings can be any shape or size and can perform a multitude of tasks.

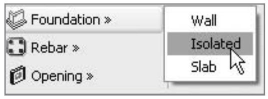
Isolated Footings

You use a different process to insert an isolated footing into the model. An isolated footing is a foundation family, not a system family. This means you can either create your own from scratch or use a predefined Revit Structure footing. For example, the step footing family in the previous exercise is actually an isolated footing. Being able to add specialty items such as this, in a manner that makes sense to the designer from a design perspective, allows the user to push Revit Structure to the limit in terms of customized functionality. Although other programs allow you to make a 3D extrusion, in Revit Structure you are piecing the model together using true components that contribute to the overall project, and in the end, to the analytical model as well. A stepped footing is similar to a spread footing with some slight differences.

Spread Footings

The most common of the isolated footings is the spread footing. Simply put, a spread footing is a concrete cube. Normally, the default out-of-the-box footing is adequate. A spread footing is inserted into a Revit Structure model in the same way a continuous footing is placed. On the Modelling tab, click the Foundation button, then select Isolated from the flyout menu, as shown in Figure 8.17.

Figure 8.17
Inserting the isolated footing

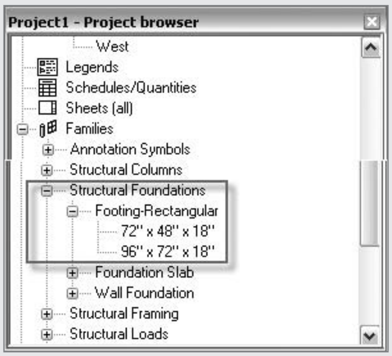


The same process is involved when placing the spread footing. On the Options bar, you will see a current footing type in the Type Selector. This, of course, is common in Revit Structure. At this point, all you need to do is select a column to place the footing under. You can also create a "free instance" by placing the footing anywhere in the model.

Real World Scenario

Controlling Content

We could begin by clicking the Element Properties button on the Options bar to make a new footing type, but let's use the Project Browser family organizer instead.



In the Structural Foundations category you will see a Footing-Rectangular category. Within this category are two default Revit Structure spread footings. By simply duplicating one of these items (right-click and choose Duplicate), you can manage these families within the project. Many times, you will know that you are going to have multiple sizes at the beginning of the job, and may want to create a few different types of footings to accommodate the project all at one time. You can accomplish this easily using this method. And remember, this is for all the families in your project, and not just footings.

The spread footing properties include Width, Length, and Thickness variables. The Thickness variable, unfortunately, does not tie in with the thickness of a strip footing that may be poured through it. Keep this in mind as you develop these footings. To access the properties, you can click the Element Properties button as you are placing the foundation; find the footing type in the Project Browser and open it from there; or select a footing from the model, right-click, and select Element Properties.

Duplicate that Family!

Be careful when you access the properties of any family. When you click the Edit/New button in the Element Properties dialog box, you are changing the properties within *all* the footings of that type. Be sure to click the Duplicate button before proceeding with dimensional changes to the element.

To place and configure a spread footing:

1. On the Modelling tab on the Design bar, click the Foundation button and choose Isolated from the flyout menu.
2. If this is the footing size you want, simply place it under a pier, pilaster, or column, or place a free instance in the model.
3. If this is not the footing size you want, click the Element Properties button on the Options bar, click Edit/New, then click Duplicate.
4. Change the size to whatever you need, and click OK to return to the model.
5. Place the footing under a pier, pilaster, or column, or place a free instance in the model.
6. As you are placing a spread footingunder a column or pier, notice the alignment lines that appear to help guide the placement of the footing, as shown in Figure 8.18. Figure 8.19 shows an isometric of the completed corner.

Figure 8.18
Alignment lines

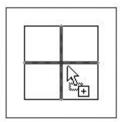
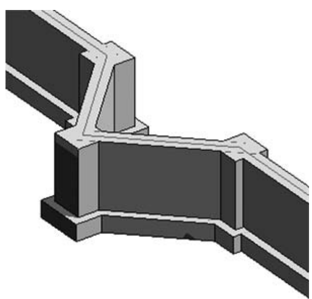


Figure 8.19
Foundations coming together



Caissons, Grade Beams, and Piles

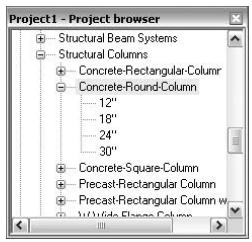
It's no fun when the geotechnical report indicates you are going to be doing some drilling. The job seems to take on a different life. The foundation is going to be special. Based on the fact that you already know how to add columns and beams, this section is going to be very familiar to you.

There are a number of caisson types. Let's examine the properties of a drilled caisson. As you can see in Figure 8.20, a caisson is actually a column. To load a "caisson" into the model, select File ➢ Load from Library ➢ Load Family. Then, in the resulting dialog box, scroll to Imperial Library/Structural/Columns/Concrete. In the Concrete directory, select `Concrete-Round-Column.rfa`.

FIGURE 8.20
Column preview

Once the family is loaded into the model, it becomes available in the Project Browser (see Figure 8.21). To turn our "column" into a "caisson," scroll to Structural Columns. Then, drill into the Concrete-Round Column category.

FIGURE 8.21
Use the Project Browser to access the Concrete-Round Column category.

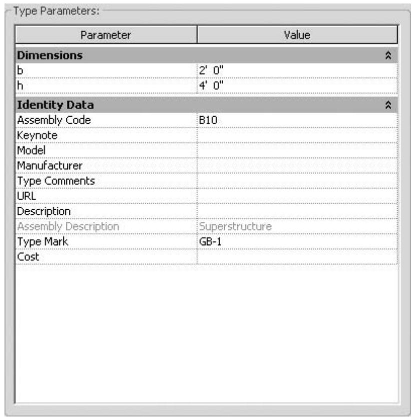


The defaults are listed. To create the caisson, right-click on the 30″ size and select Duplicate. Rename the new caisson to **`Caisson type C-1 (36" dia.)`**. With the new caisson in place, double-click on it to gain access to its properties (see Figure 8.22). In the properties, you can change the size as well as some other important items that have to do with the tagging of the members. As mentioned at the beginning of this chapter, drawing aesthetics are usually of great importance. Tags, text, and hidden-line configurations all contribute. But it is at this level where we can allocate a member to be automatically tagged at a later time. It is nice to be able to achieve your company's drafting standards while using a Revit Structure database and the BIM approach. In the Type Mark category in the Element Properties dialog box, add the Caisson type mark of C-1.

Once you have created the caisson, it is time to insert it into the model. On the Basics tab on the Design bar, click the Structural Column button to place the caisson.

Certain elevation considerations pertain to an augured type of foundation member—in particular, the depth. You can handle this in one of two ways. One is to actually create a level for the bottom of the caisson. This approach works if all the caissons are to be drilled to the same depth. If the depths will vary—which is usually the case—you can simply set Depth to Unconnected and specify an increment. To put it simply, it would be nice to specify a bottom-of-caisson level, but Revit Structure gives you the flexibility to do both.

Figure 8.22
The new caisson's properties



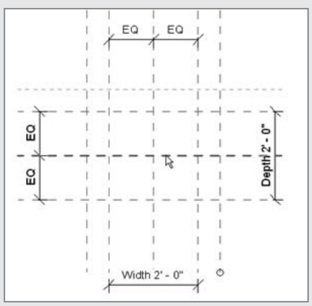
Type Parameters:

Parameter	Value
Dimensions	
b	2' 0"
h	4' 0"
Identity Data	
Assembly Code	B10
Keynote	
Model	
Manufacturer	
Type Comments	
URL	
Description	
Assembly Description	Superstructure
Type Mark	GB-1
Cost	

Exercise: Creating Specialized Caissons

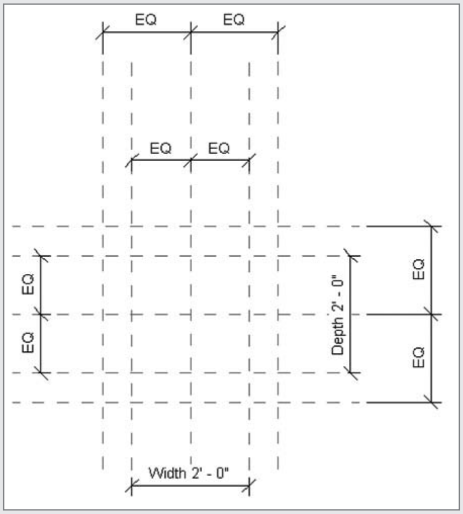
It's all in the family…literally. This is what drives Revit Structure. Although the hard-core families are saved for the final chapter of this book, it is a good idea to study the family concept here as it pertains to caissons. A structure we completed in Revit Structure was a 1950s-era parking garage in which bell caissons were used. We needed a bell caisson with a flared top. Yes, there are a few types to choose from in the stock Revit Structure library, but we needed customized functionality! We decided to make a quick bell caisson using the following process. Keep in mind that this is a fast example of an advanced topic. If it proves to be somewhat tricky, we recommend that you skip ahead to the last chapter and explore the concepts of creating a family.

1. Choose File ➢ New ➢ Family. Then select `Structural Column.rfa`.
   The default view will be in plan. We need to add some reference planes.
2. On the Family tab (this replaces the Design bar to the left of the screen), click the Ref Plane button. Reference planes are crucial to the creation of families.
3. On the Options bar, click the Pick Lines button, and specify an offset of 1´-6˝. In plan view, pick the center reference planes and offset them to the left, right, top, and bottom. (See the following graphic.)

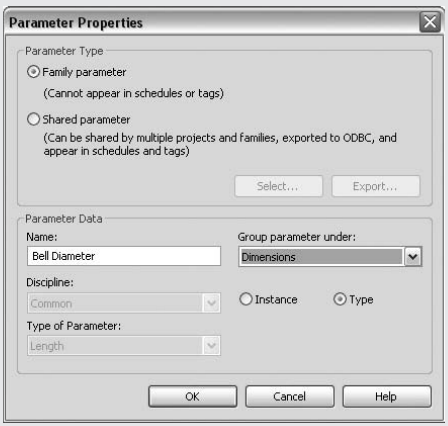


4. On the Family bar, click the Dimension button.

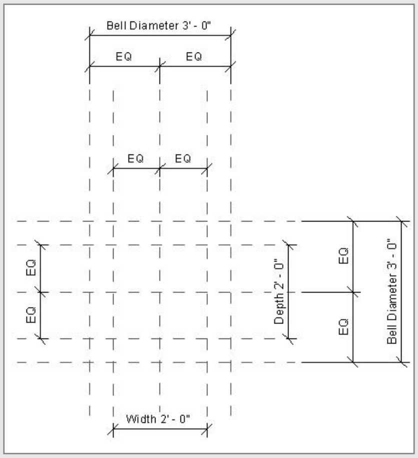
5. Create equal dimensions by selecting the new reference planes. Make them equal based on the two center reference planes. They should appear similar to the equal dimensions that have already been created in the plan. Remember that once you create these dimensions, you need to select the blue EQ icon that appears. If the dimensions still reads as an increment, they will not constrain the family to center itself on the centerlines.



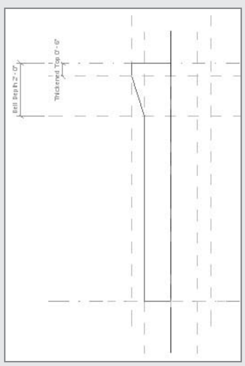
6. Add two more dimensions across the top and the side. These dimensions should be from the far left to the far right, and from the top to the bottom reference planes. They are the total of the EQ dimensions, and should read 3´-0˝.
7. Select these dimensions, and on the Options bar, choose Add Parameter from the Label drop-down menu.
8. In the Parameter Properties dialog box, name the parameter **`Bell Diameter`**.
9. From the Group Parameter Under drop-down menu, choose Dimensions.
10. Click the Type option.



11. Click OK.



12. In the Project Browser, double-click on the Front elevation. You will see an upper and a lower reference level. Both are important.
13. Select the Ref Plane command and offset a reference plane down from the upper level 2´-0˝.
14. Select the Ref Plane command again, and offset a reference plane down 6˝.
15. Select the Dimension command, and dimension both reference planes. Be sure that these are separate dimensions, both starting from the upper level.
16. Select the 2´-0˝ dimension, and add a label to it. Name the parameter **`Bell Depth`**, choose Dimensions from the Group Parameter Under drop-down menu, and select the Type option.
17. Select the 6˝ dimension, and add a label to it. Call the parameter **`Thickened Top`**, choose Dimensions from the Group Parameter Under drop-down menu, and select the Type option.
18. In the Family bar, click the Solid Form button, and then select Solid Revolve from the list.
19. On the Options bar, click the Pick Lines button and turn on the Lock toggle. Proceed to select the lines only to the right, including the center of the column. You will have to draw the angled line that connects the Bell Depth to the Thickened Top dimension.
20. Click the Axis button and select the center reference plane.



21. Click Finish Sketch. If Revit Structure tells you there are issues with the lines, pay attention to the orange lines that are displayed. These lines should form a continuous loop with no overlapping lines and gaps. This process takes practice and may take an equal amount of patience.

22. Go to a 3D view.

23. Click the Family Types button on the Family bar and change some of the parameters for Bell Depth. The revolved mass should change to reflect the incremental changes.

24. While in the Family Types dialog box, click the Add button in the Parameters field in the lower-right corner.

25. Name the parameter **`Material`**.

26. For the Type, specify Material.

27. Choose Materials and Finishes from the Group Parameter Under drop-down menu.

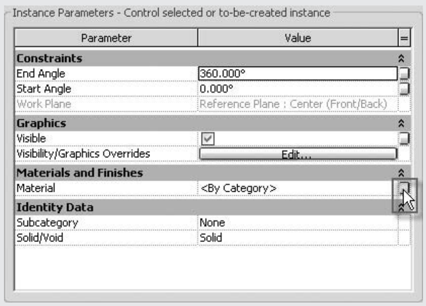
28. Select the Type option.

29. Click OK twice to return to the model.

30. Select the caisson.

31. On the Options bar, click the Element Properties button.

32. Under Graphics in the Element Properties dialog box is a row named Visible. You will see a check mark. In the far-right corner you'll see a little button. Click it.



33. Select the Material parameter from the dialog box. This allows the material to be changed in the project as opposed to forcing a material to the family. You will get more flexibility this way.

34. Save the file as **`Bell Caisson`** in the Columns library directory.

35. Click the Load into Projects button on the Family bar.

36. In the Basics tab on the Design bar, click Structural Column. Select Bell Caisson from the Type Selector drop-down menu.

37. Add the caisson to the project. You can now configure and create new instances of this family just as you did with the other families we've studied up to this point.

Grade Beams

Adding a grade beam to a model is almost the same as adding a concrete framing member. The only real differences are the physical size, justification, and labeling considerations. You may start to notice some redundancy in the methods and concepts of foundations as it relates to the rest of the model. Revit Structure is built to work the same whether you are on the roof or at grade.

Grade beams are beams, so to configure a new beam, drill into the Project Browser (no pun intended) and locate Structural Framing, then Concrete Rectangular Beam. In this category, you will find a couple of different concrete beams. Right-click one of them and duplicate the member. Rename the new beam to **`Grade Beam GB-1 (24x48)`**. Double-click on the new beam to access the Element Properties dialog box. In the Type parameters of the new grade beam, change the *b* value to 2′-0″ and the *h* value to 4′-0″. Add the Type Mark of GB-1.

To add a grade beam to the model, you must deal with the caissons and any element that the grade beam bears on. For this example, a 48″- deep grade beam is being placed in the model. Before the grade beam is to be drawn in, the caissons need to receive an offset of –48″. By selecting all of the caissons already in place and clicking the Element Properties button, you can set the top offset to –48″. With the grade beams in place at the proper elevations, the next step, adding the slab, will tie the system together. The plan view will show the hidden lines of the grade beam/caisson system below, and will produce sections that are close to being ready to put on a drawing sheet.

To configure and add a grade beam:

1. Set all caisson and bearing members to receive an offset consistent with the thickness of the grade beam.
2. In the Project Browser, find Structural Framing, then Concrete Rectangular Beam. Copy one of the predefined beams, and duplicate it.
3. Change the Type parameters of the new beam. Also, change the name to one that reflects how your firm likes to label grade beams. Be sure to add a Type Mark identifier such as GB-1.
4. On the Basics tab on the Design bar, click the Beam command.
5. Find the Grade Beam in the Type Selector.
6. As you are placing the beam, click the Element Properties button on the Options bar, and change the Z-Direction Justification to Bottom.
7. Create the grade beam in the model, as shown in Figure 8.23.

Now that the structure of the foundation has been taken care of, it is time to move on to placing concrete slabs.

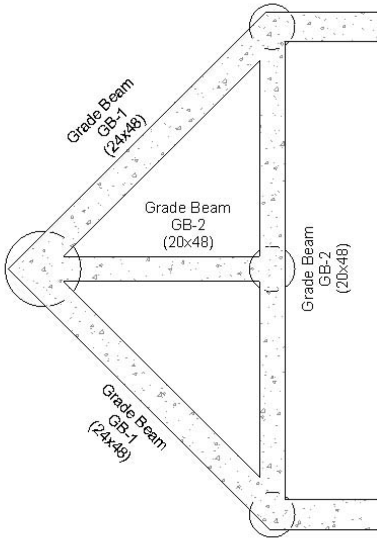


FIGURE 8.23
Grade beams

Foundation Slabs

Slabs and floors are different. In an actual floor, we specify items such as steel decking, and perhaps a floor finish here and there. A slab bears on earth with a vapor retarder in between. Plain and simple. Of course, reinforcing considerations are different, and we will address viewing the cover settings, but to create a slab, we are still in the Foundation mind-set. To carry out the placement of a foundation slab, select the Modelling tab on the Design bar, and click the Foundation button. Then choose Slab from the submenu, as shown in Figure 8.24.

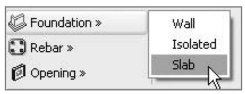


FIGURE 8.24
Adding the slab

The default Revit Structure slabs are 6″, 8″, 10″, or 12″ of concrete. That is basically it. It would be nice to have a slab complete with a vapor retarder. In the Project Browser, you can find all of the structural slabs that are currently configured for this model under Structural Foundations ➢ Foundation Slab. In this case, it is convenient to start with one of the existing slabs and double-click to gain access to the Type Parameters dialog box. In this dialog box you'll find the Structure category. By drilling into this category, you can add layers to the slab, as shown in Figure 8.25. The way this is set up is determined by the structural component surrounded by nonstructural elements that are separated by the core boundary. We can add items to the top or the bottom of the slab as we wish without affecting the structure.

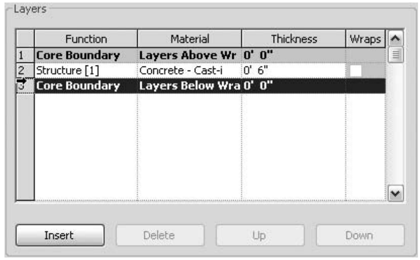


FIGURE 8.25
Slab layers

As you can see in Figure 8.26, by selecting the bottom Core Boundary row and clicking the Insert button, we can add another row. If the new row is added inside the core boundary, you can click the Move Down button to move the vapor retarder out of the core. Name the function **`Thermal/Air Layer`**. Click in the Material column, and a builder button (the tiny button with an ellipsis) shows up. Click that, and you can find the material called Vapor/Moisture Barriers – Vapor Retarder. Change the thickness to ⅛″ (about 3 millimeters). Change the core concrete thickness to 5⅞″ (this avoids confusion with the overall thickness).

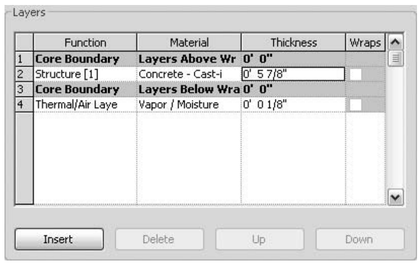


FIGURE 8.26
Adding the thickness

Once the foundation is completed, it is time to add it to the model. From the Modelling tab on the Design bar, select Foundation ➢ Slab. The Design bar turns into the Sketch bar. Make sure the Pick Supports option is checked, and then you can select the perimeter. Again, you must have no gaps or overlapping lines. Sketch the perimeter as shown in Figure 8.27.

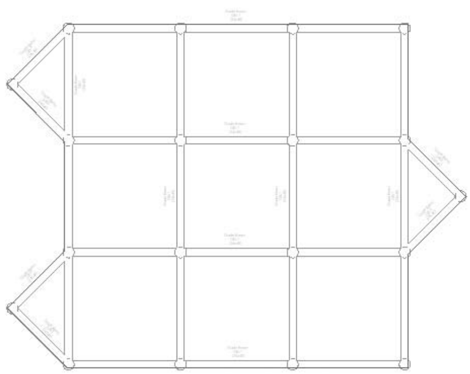


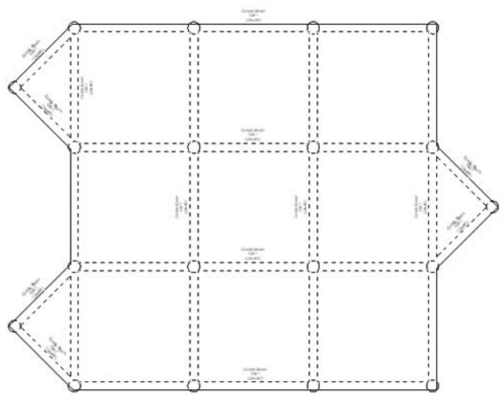
FIGURE 8.27
Sketching the perimeter

To manage and add slabs to a foundation, follow these steps:

1. In the Project Browser, drill down to Structural Foundations, then select Structural Slabs. In this category, you will find several predefined sizes. Select the one with the thickness you want, and double-click it to access the Type Parameters dialog box.
2. Find the Structure row, and click the Edit button.
3. In the Layers dialog box, insert a new row. This row will be a Thermal/Air barrier, and will consist of a material called Vapor/Moisture Barriers – Vapor Retarder. Make the thickness ⅛″.
4. You can now add this slab to the model by either dragging it in from the Project Browser, or by selecting the Modelling tab on the Design bar and choosing Foundation ➤ Slab.
5. Click the Properties button and set the Height Offset From Level to 6″.
6. Select the outside perimeter using the Lines command.
7. Once you are done, click the Finish Sketch button.

As you can see in Figure 8.28, the nice thing about the graphics involved with Revit Structure foundations is the hidden line scheme. It just makes sense that items that have become obscured by other items are going to displayed as hidden lines. By completing the slab, it is time to make sure the graphics appear the way you want. Check the Visibility/Graphic Overrides dialog box options, the View Control bar, and the properties of the elements being displayed. At this level, you are ready to start adding other items such as piles and perhaps a site.

FIGURE 8.28
Hidden line composition



Piles and Pile Caps

Revit has some nice families already created for piles and pile caps, and adding them to the model is becoming old hat. Piles and pile caps are considered foundation families. Although it would naturally seem as though they would be added to the model in the traditional beam/column method, you load the pile and pile cap family using the Modelling tab on the Design bar. For this example, we will load a pile cap grouped with the associated piles, as shown in Figure 8.29. Choose File ➤ Load from Library ➤ Load Family. In the resulting dialog box, browse to Structural, then Foundations. Load the family called `Pile Cap-3 Pile.rfa`.

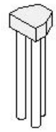
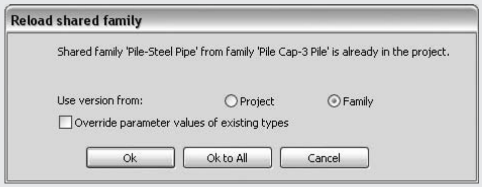


FIGURE 8.29
A pile group

OVERWRITING A FAMILY

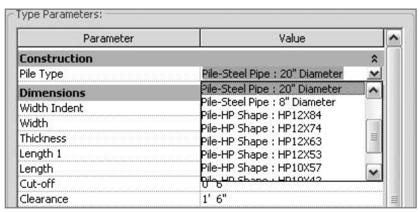
Note that as you load families into the model, occasionally you will find that a family is already loaded into the model. Revit Structure allows you to either leave the preexisting family intact or overwrite the existing family with the new one. Be very careful here. If you check the option that says Override Parameter Values of Existing Types, you will wipe out any type parameter for that family that has been changed. This can be good, or it can be disastrous. We recommend you click OK but leave the Override toggle deselected, as shown here:



With the new family safely loaded in the model, you can choose the Modelling tab, and click Foundation ➢ Isolated. You add the pile groups as you would a column or spread footing.

The Element Properties dialog box of a pile family provides quite a few choices in terms of configurations. You can set up the reinforcing cover in the cap as well as the dimensions. Also, in Type Parameters, you can choose which piles this family is grouping with the cap, as shown in Figure 8.30. By default, it is a tubular member, but you can switch it to an H shape. If you find that the H shapes are not available, choose File ➢ Load from Library ➢ Load Family. Select Structural, then Foundations, and locate the file `Pile-HP Shape.rfa`. A separate dialog box opens that allows you to choose from a catalog. It does not hurt to load more family types than you need.

FIGURE 8.30
Type Parameters for a pile



As you can see, once you have mastered adding a column to a model, you have mastered adding a pile group. Redundancy is a good thing.

To add pile groups to your model:

1. Select File ➤ Load from Library ➤ Load Family. Browse to Structural, then Foundations, and locate the pile or pile group.
2. Once you've loaded the pile into the model, configure the Instance and Type parameters to suit your situation. If needed, create a new type by clicking the Duplicate button.
3. Click the Modelling tab on the Design bar, and select Foundation ➤ Isolated.
4. Add the piles to the model.

Another consideration is the placement of an elevator pit. Although it has basically been created, it does need to be specifically addressed.

Elevator Pits

There is always going to be a specialty item in a foundation. Elevator pits are certainly at the top of the list. An elevator pit combines quite a few elements of Revit Structure. Not that creating an elevator pit is difficult—quite the contrary—but it usually consists of the results of work that has been done in other areas. For instance, an elevator pit is at the bottom of an elevator shaft, right? Right. So you might have a shaft opening. This shaft opening will cut through the foundation slab. By the time we get to the elevator pit, a slab is in place and has been cut to the correct dimensions. Also, the upper masonry walls that define the shaft are in place. All we need are four additional concrete walls, a big chunk of concrete, and some notes, and we are in business!

To get started, let's address the items we haven't looked at in this chapter: masonry walls and shaft openings. An elevator shaft is roughly 10´-0˝ clear to the inside dimensions. The masonry walls themselves are 8˝, 10˝, or 12˝ Concrete Masonry Units (CMUs). There will be no floor structure in this opening, but there can be framing. This will be taken care of by the shaft opening. In plan view, add four masonry walls with an inside opening of 10´-0˝ (we recommend that you set up reference planes as a guideline). These walls will be constrained at the top usually by a penthouse level (if there is a hoist beam) or at least by the roof. The masonry walls will stop at the slab on grade or at the basement level. The area from that level down to the top of the footing level will be supported by concrete bearing walls. These bearing walls will rest on a thickened concrete pad. There may be a sump taken from this area. The sump can be modeled, or it can be drawn in. Basically, you are drawing four walls that stop at the first floor and drawing four concrete walls underneath it.

To create an elevator shaft:

1. Draw four CMU walls in plan. Set the base constraint to the level the structural slab is on.
2. Once the walls are drawn, select all four of them. Click the Attach button on the Options bar. Attach the base to the floor slab.
3. Cut a section through the elevator pit. You will eventually need it for the construction documents, and it will help you model the pit.
4. Add four 12˝ concrete foundation walls from your first-floor level to the top of the footing level. As shown in Figure 8.31, in the Element Properties dialog box, set the top offset to the thickness of your slab. This will ensure that the masonry wall is bearing directly on the top of the foundation wall.

FIGURE 8.31
Set the top offset to the thickness of your slab.

Parameter	Value
Constraints	
Location Line	Wall Centerline
Base Constraint	T.O. Footing
Base Offset	0' 0"
Base is Attached	
Base Extension Distance	0' 0"
Top Constraint	Up to level: Level 1
Unconnected Height	4' 6"
Top Offset	0' 6"

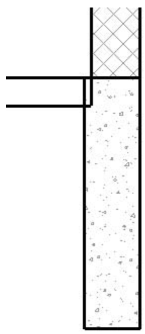
5. On the Modelling tab on the Design bar, click the Opening button, then select Shaft Opening. Draw a rectangle around the exterior of the masonry walls. In the Shaft Properties dialog box, set Base Constraint to T.O. Footing and Top Constraint to the top of the elevator shaft, as shown in Figure 8.32.

FIGURE 8.32
Use these Base and Top Constraint settings.

Parameter	Value
Constraints	
Top Offset	0' 0"
Base Offset	0' 0"
Unconnected Height	24' 0"
Base Constraint	T.O. Footing
Top Constraint	Up to level: Level 2

Once you have completed these steps, you will have something that looks like Figure 8.33. Be sure there is a continuous bearing extending from the CMU shaft walls to the concrete foundation walls. The challenge is to create a ledge in which the slab can rest on the concrete wall. You can accomplish this using the Join command. Wherever there is overlapping concrete, the Join command can clean up the extra concrete lines and create the bearing ledge, as Figure 8.33 shows.

FIGURE 8.33
The ledge before the join



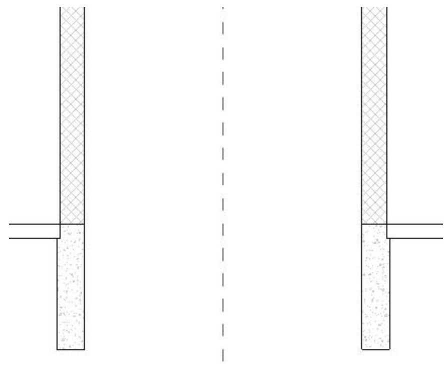
To join the ledge to the slab, follow these steps:

1. Click the Join Geometry button on the Tools toolbar, as shown in Figure 8.34.

FIGURE 8.34
Click the Join Geometry button.

2. On the Options bar, elect the Multiple Joins option.
3. Select the concrete slab.
4. Select the two walls. The foundation wall now has a bearing ledge that the slab is sitting on, as shown in Figure 8.35.

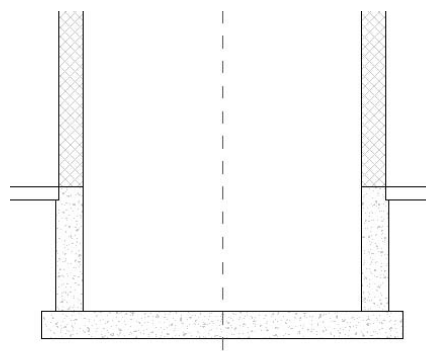
FIGURE 8.35
Coming together



5. Go to the T.O. Footing plan view.
6. On the Modelling tab on the Design bar, click Foundation ➢ Isolated. In the Type Selector drop-down, select one of the square bearing footings.
7. Click the Element Properties button, and click Edit/New. Click Duplicate in the resulting dialog box.
8. Name the new footing type **13' x 13' x 12"**.
9. Change the width and depth to 13´-0˝.
10. Insert this big footing under your pit. (If you had put the recommended reference plane in, you would be snapping to it right now.)

Figure 8.36 shows the completed pit.

FIGURE 8.36
The modeled pit



There is still a lot of work to be done. These tasks will be addressed in separate chapters. The intent of this procedure is to get as much modeled as possible. Now, when it comes time for notating, detailing, and reinforcing, it is going to make the process that much easier and as accurate as possible. Now when we add a site to the model, we will be able to use the cross sections in the details along with our model.

Modeling an Imported Site

Earth is always a consideration. At some point, our foundation is being imposed on by the pressures of Mother Earth. Although we cannot calculate lateral forces, we can at least import the geometry from Autodesk's Civil 3D or other CAD applications. The procedure starts by having a good, clean CAD file. In Revit Structure, there is a Site tab on the Design bar. If you do not see the Site tab, right-click on any tab and select Site from the context menu, as shown in Figure 8.37.

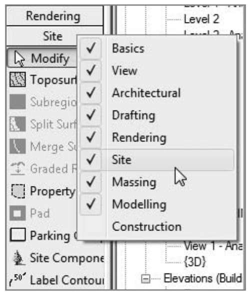


FIGURE 8.37
Adding the Site tab

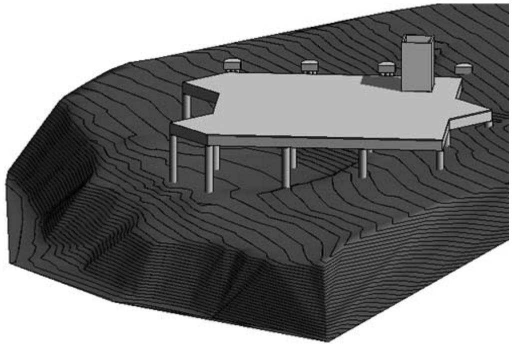
The File menu contains an Import/Link option. From the flyout menu, you can choose CAD Formats. Revit Structure will open the Browse for CAD Files dialog box. You can then find the CAD file. Go slowly here, because at the bottom of this dialog box are a few choices. Linking the file is a good idea. With the CAD file linked, the orientation of the CAD file can be coordinated with the orientation of the site in Revit Structure. With the link choice toggled on, click OK, and the site appears in your model. There may be some scaling issues. Normally the site is 12 times too small. In the Import dialog box, be sure the scaling is set to something familiar if the site is not coming in sized as expected.

Visibility Settings

Once you have imported the site, you can then drape a topographical surface over the top of the contours, as shown in Figure 8.38. The problem is, Revit Structure is not ready for a site plan just yet. You will need to tell Revit, "Hey, we have some site stuff." By opening the Visibility/Graphic Overrides dialog box (press VG), you can scroll through the list and find Topography. But this

category is not shown in Revit Structure by default. At the bottom of the dialog box, you must click Show Categories from All Disciplines. Once you do, you can select the Topography option.

FIGURE 8.38
Importing a Civil 3D site



On the Site tab, click the Toposurface button. In Sketch mode for the Toposurface, select Use Imported ➢ Imported Instance. Once you select the imported CAD file, a Layers dialog box appears. If you have imported a clean model, this will be easy. If not, there may be some searching to do. Usually there will be "Cont…" layers for contours to select. When you select only these layers, Revit Structure will know to triangulate the surface based on the geometry of these lines.

One other issue with adding a site to your model is that there is going to be earth indiscriminately "spilling" into your building. Or, the structure may be floating above the site. In the next segment, we will discuss how to displace this earth using pads.

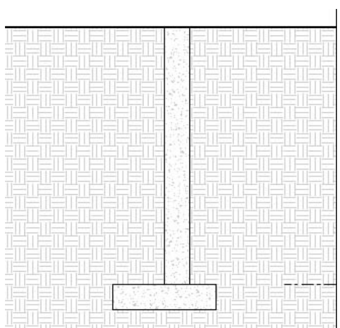
Pads

As shown in Figure 8.39, the earth breeches our structure. Therefore we need a method of displacement. We achieve this through the use of pads. Pads are added to a model exactly as a slab would be, only Revit knows to either cut or fill topography to meet a pad. The most useful application for a pad is right on top of the footings.

The application for a pad is as follows:

1. On the Site tab, click the Pad button.
2. In your T.O. Footing level, sketch the pad to the extents that you would have the earth after backfill—usually to the outside face of the concrete.
3. Once you finish the sketch, the earth will be displaced by the pad.
4. If you are in a section, right-click the pad and select Hide in View.

FIGURE 8.39
Before the pad



Now that we've placed the pad in the model and hidden it in the section view, the earth serves two purposes. One is for the modeling aspects, and the other is for the detailing. It is nice to not have to draw and hatch the earth in every detail, as indicated in Figure 8.40.

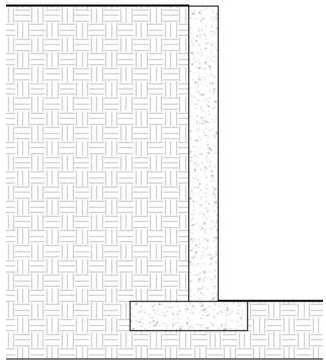


FIGURE 8.40
After the pad

The Bottom Line

Create strip footings. This chapter illustrated how to best add continuous footings to the model. Strip footings are placed underneath a wall in a continuous manner. The walls that bear on these footings will dictate the location and length.

Master It There are two ways to create additional strip footing families in Revit Structure. One is to click the Element Properties button on the Options bar as you are placing the footing. What is the other way of creating a bearing footing as a wall foundation?

Create footing steps. A stepped footing occurs in Revit Structure when you edit the profile of a wall to step into a graded area of a site. Additional families need to be added to complete the sequence. In this chapter, you learned how to modify a wall, and what actions to take to create a stepped footing condition.

Master It A footing will step with the wall, if the profile of the wall has been modified, but the step will simply drop with no angled bulkhead.

A. How do you edit the profile of a wall to step it?

B. How can you add a stepped footing family?

Create and add foundation slabs. Foundation slabs are built in Revit Structure very literally. Slabs consist of material "layers," which in turn provide you with an overall slab thickness. In this chapter, you learned how to find a predefined foundation slab and modify it to suit your needs.

Master It How can a slab be modified to include a vapor barrier?

Create grade beams. You place a grade beam in Revit Structure in the same way as you would a steel beam. Grade beams can be of any size and thickness. In this chapter, you learned the procedure for adding a grade beam to your model.

> **Master It** Grade beams are added to the model as part of a foundation, but grade beams are added as an actual beam. What is the process for this?

Create elevator pits. Elevator pits are not an actual function within Revit Structure, but a compilation of structural items included in the foundation. In this chapter, you learned how to add an elevator pit to your model by combining several items.

> **Master It** Which elevator pits are "formed" when a collection of elements are brought together, and how?

Part 3

Documenting Your Structural Model

Chapter 9

Model Documentation

Few structural engineering firms can afford to make a three-dimensional model for its own sake. One of the main and most important strengths of Revit Structure involves its ability to create construction documents in addition to the 3D physical and analytical models. Not all modeling software can do that. In this chapter, you will focus on those documentation features and the challenge that lies in store for you as you advance your model through the design process toward completion of your construction documents.

This chapter on documentation features in Revit Structure centers on functions that are found on the Drafting tab of the Design bar, as well as the creation of a typical details library. You will examine the methods necessary to annotate your views, whether it is a plan, section, elevation, or other view type, and complete the necessary documentation.

In this chapter you will learn to:

- Add datum elements to your detail and section views
- Add annotation elements such as text, tags, and symbols
- Add detailing elements such as detailing lines and filled regions
- Create a typical details library

Drafting Tools

A lot of the features you need to finish your details that have been cut from the model, or developed as simple two-dimensional drafting views, will be found on the Drafting tab on the Design bar (see Figure 9.1). The Drafting tab contains several groups of related drafting commands:

- Datum commands such as dimensions, grids, reference planes, and scope boxes
- Annotation commands such as Tag All Not Tagged, Keynote, and Span direction tags, as well as Text for notations
- Detailing commands such as Detail Lines, Filled Regions, and Repeating Details

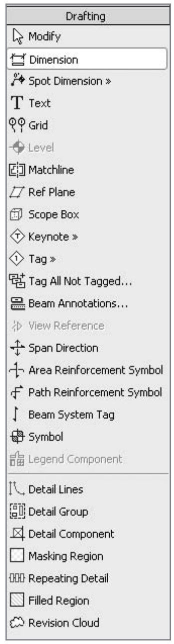


FIGURE 9.1
The Drafting tab on the Design bar

That is how you will study them in this chapter, as you learn how to take an undocumented view and complete it for your construction document set. It is important to remember that, like model elements, these elements are also families that can act parametrically. For example, the Text family can have many types, such as a 3⁄32″ Arial, or 1⁄4″ Times New Roman. Once you start developing these text types that correspond to the styles your company uses, you can add them to your project template file. Then when you begin a new project, the types will be ready to go.

Datum Elements

Datum elements aid in the precise location of elements in your model. For a structural drawing, it is imperative to have a well-dimensioned project with all major elements properly located, as well as an organized grid system. Next is an overview of the various datum elements and how they aid in the documentation of your project.

Dimensions

The Dimension command is located on the Basics and View tabs as well as the Drafting tab on the Design bar. You can also access the command on the Drafting tab of the menu bar. Once you click the Dimensions tab, the Options bar will display various tools for you to use to format and place them (see Figure 9.2).

FIGURE 9.2
The Options bar for dimensions

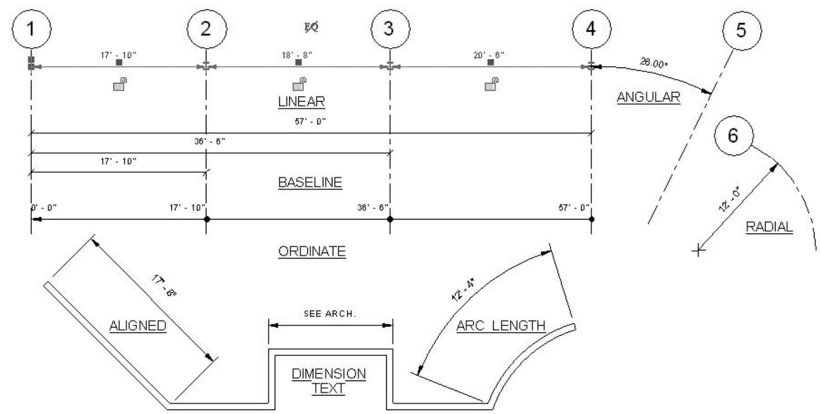
In the Type Selector are different family types of dimension styles for you to use, or you can create your own types. If you click the Element Properties button to the right of the Type Selector, the Element Properties dialog box for dimensions appears. There you can create or edit the graphical elements of the dimension styles as well as the lines and tick or arrow marks, in addition to formatting the dimension text.

To the right of the Element Properties button you choose among various forms of dimensions:

- Aligned or Linear
- Angular
- Radial
- Arc Length

Figure 9.3 shows examples of these various dimension types that you can use in your project. Dimension styles can be created as linear, baseline, or ordinance styles. Notice that arrows, tick marks, and dots (as well as others not shown) can be configured as dimension endpoint symbols depending on your preference.

FIGURE 9.3
Dimension types



The following sections will familiarize you with the usage of dimensions, how they function, how they are placed, and how the dimension text string can be adapted to show a note instead of the dimension value.

Constraints: Locks and Anchors

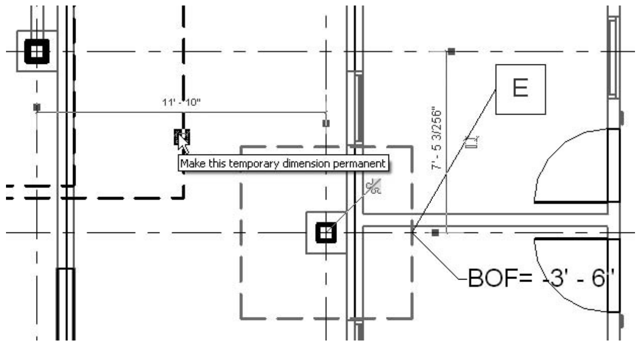
You have been learning that Revit Structure is a constraints-based system in which relationships between elements are maintained as the model flexes to new positions during the design process. Dimensions are one of the main ways to control this movement. In Figure 9.3 the linear dimension is highlighted and two noteworthy controls are displayed. Under the linear dimension line are locks that are currently open. By clicking the locks closed, you will prevent that dimension from being changed. Above the linear dimension line, the EQ control is displayed. If you click the EQ symbol, the dimensions will be equalized and each will display an EQ value in the dimension string. This is a helpful command that allows you to easily equalize the relative locations

of elements. Dimension anchors, represented by an anchor symbol at the start of an equalized dimension string, are used in a multisegmented EQ constrained dimension to indicate which element remains stationary when the elements change location.

Listening, Temporary, and Permanent Dimensions

As you place and locate your model elements, you will be using dimensions long before you need them for your documentation. As you place elements in your model, temporary dimensions begin appearing dynamically as you are working (see Figure 9.4) to help you place the element precisely. By clicking on the dimension control symbol next to the dimension, you can automatically add the dimension to your view and make it permanent.

Figure 9.4
Making temporary dimensions permanent



Another important concept is what are known as *listening* dimensions. As you start to add an element to your view, such as a line or arc, after the initial click to place it, you can type in the value for the length you want in the temporary dimension that appears in order to place the second point.

Real World Scenario

Don't Forget Your Documentation

It is easy to be caught up in building a fabulous virtual model when suddenly the boss requests a current document set of all the sheets of plans, elevations, and sections so he can show your progress to the client. You have been so absorbed in model building that you forgot to develop the views you needed for your documents. To your boss, it appears you have not done much work, even though you show him the cool 3D views of what you have been building on the computer.

The point here is that you need to start any job with this in mind and document your model from the very beginning. Get the title block done and loaded into the project so sheets can be created. Begin importing your typical details into drafting views and placing them on sheets. Make a sheet of 3D views as a great source of information for the design team and client, even in schematics. As you cut working sections, keep in mind whether they might be worthy enough to be used in the documentation. If so, drag them onto a sheet.

Dimension Placement

Placing dimensions is as easy as clicking on successive elements in the view. But there are also several options for you to use for dimensioning walls. On the Options bar you will find the Prefer option, which allows you to dimension to wall location lines, such as the center of wall cores, or to wall faces. Just select one of these options to enable it as you begin placing the dimension strings. You can also select Individual References as Entire Walls as a way to dimension your elements. If you select Entire Walls, Revit Structure enables the Options button and you can configure it to automatically do such things as dimension to openings in the wall. With one click, you can create the entire string of dimensions.

After you place the dimension, you can move the whole string by highlighting it and then dragging it to a new position. You can move the text string and lines together or independently by selecting and dragging them. Another important new capability allows you to select intersection points of lines and grids as dimension points for the string.

Dimension Text

As you are documenting your project, you will undoubtedly come upon cases where you want to show a dimension string with a text note inserted onto it instead of a length or angle value. This capability has just been added to the 2009 version of Revit Structure and comes in very handy. To use this feature:

1. Highlight an existing text string.
2. Click on the dimension text, which then displays the Dimension Text dialog box (see Figure 9.5).
3. Click the Replace with Text radio button.
4. Type the text string you want to use in the adjacent edit box.
5. Click OK.

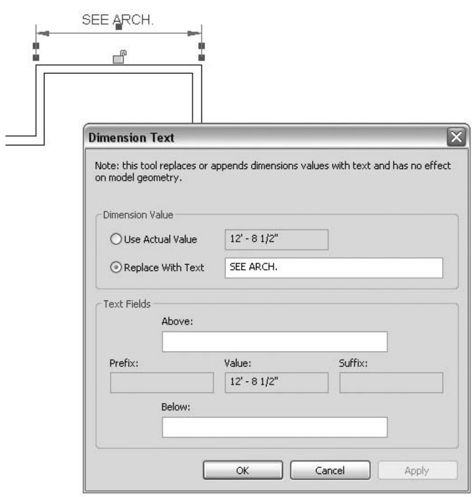


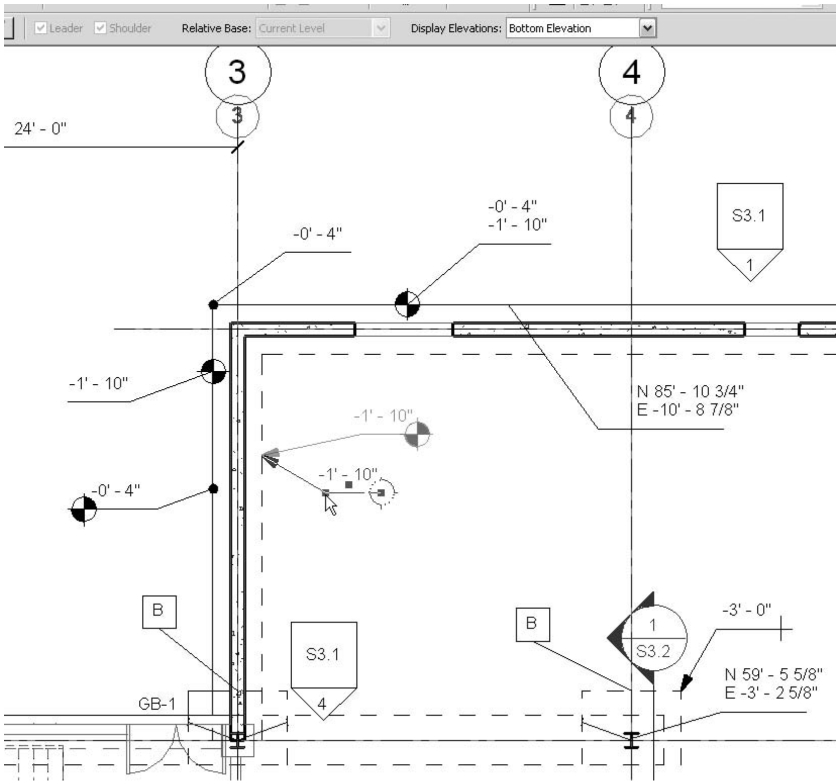
Figure 9.5
The Dimension Text dialog box

You now have a basic idea of how dimensions work. The next section will introduce you to adding spot dimensions and coordinates into your project views.

Spot Dimensions and Coordinates

Spot Dimensions come in two forms: spot and spot coordinates. Each one can be configured in a variety of ways, using leaders with or without shoulders (horizontal lines). There are also many leader end options you can choose from. If space gets tight on your plan or section, you can drag the leader to a new position (see Figure 9.6). The leader symbol will also rotate with objects.

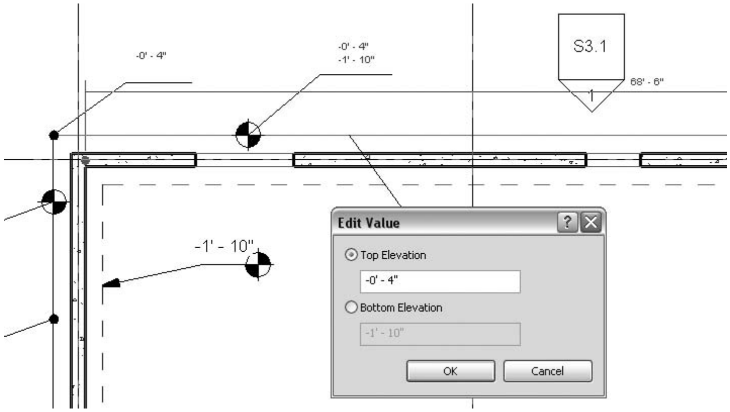
FIGURE 9.6 The many spot elevation and coordinate family types



Spot coordinates can display the top elevation, the bottom elevation, or both for a particular model element and are especially helpful in documenting concrete monolithic buildings. Once you place the spot dimension, you can use it to change the position of the object later simply by changing its value (see Figure 9.7). To edit the top of footing, follow these steps:

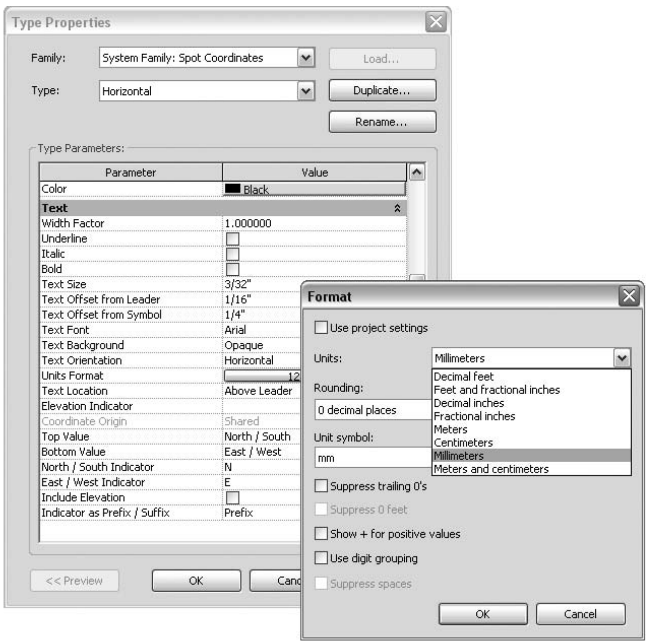
1. Highlight the object.
2. Click on the spot elevation text you want to change.
3. In the End Value dialog box that appears, adjust the elevation values.
4. Click OK.

FIGURE 9.7
Changing the wall footing elevation using a spot elevation



The spot coordinates cannot be edited in plan view once placed. The display is controlled by default through the units format, or can be overridden (see Figure 9.8). They can be shown in a variety of formats, from metric to decimal to feet and inches, with lots of options to suit just about everyone.

FIGURE 9.8
Overriding the spot coordinate format



Grids

Grids are an essential feature to understand. They are the basic horizontal constraint system in your model. Model elements such as columns and girders can be anchored to them. When a gridline moves, those elements anchored to it also move, providing your model the ability to flex as design changes occur. They are automatically generated in any view in which they intersect, saving you a lot of detailing time. That can be a two-edged sword, though, because managing them in the multitude of views that you use for your documents can be a frustrating task. Revit Structure has features to control the display in order to make managing them easier. And do not forget that the grid head can be configured as a circle, hexagon, or diamond shape, however you might prefer.

You will now examine their basic control features, how to propagate grid layouts among various views, and how to use scope boxes to control their extents.

Basic Control Features

Figure 9.9 shows an existing gridline and bubble with the controls displayed. A check box allows the grid bubble to be turned off or on at each end depending on how you want it shown in your view. Clicking the elbow symbol jogs the gridline so that the grid head can be dragged to a new position without moving the gridline itself, an important capability when you have numerous gridlines close together. By clicking on the grid text, you can change it on the fly. As you place the grids, Revit Structure numbers or letters them sequentially to make their annotation more automatic.

Notice in Figure 9.9 that the left grid head has a 2D control below it, while the right grid head has a 3D control above it. These correspond to a filled circle for 2D and an open circle for 3D. When you enable the 2D option, the grid length becomes specific to that view only. With the 3D bubble enabled, changing the gridline end in one view will also change it in every other view in which it intersects. As stated earlier, you can spend endless time moving grids back and forth on your views, so be aware of how this feature works.

Once you have placed grids on one view, say your first floor and foundation plan, and you have configured and located the grid bubbles to your liking, you will want to automatically duplicate that grid layout in other views. If you go to those other views the grids will be there, but they may not be configured exactly like the foundation plan. To accomplish this goal, you use the Propagate Extents option, as follows:

1. Highlight an existing grid in a plan view and then click the Propagate Extents button on the Options bar.
2. When the Propagate Datum Extent dialog box appears, check the views where you want to duplicate the grid display arrangement.
3. Click OK.

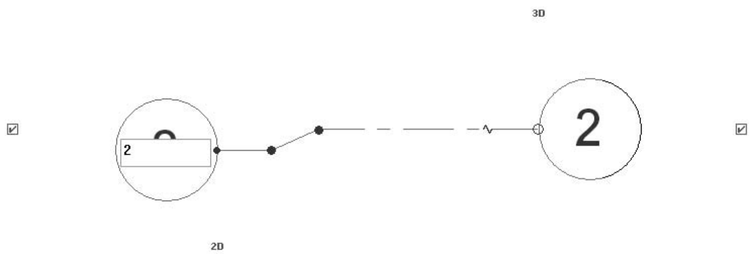


FIGURE 9.9
A basic grid element with controls displayed

> **THE GRID IS NOT SHOWING IN ALL VIEWS! WHAT NOW?**
>
> On quite a few occasions you may find that views are missing grids and levels that should be displayed. Grids and levels must intersect a view to display automatically, but even then they may not. What do you do? Well, don't get flustered and start faking them with 2D centerlines. What you need to do is find a view where the grid or level is showing. Highlight the gridline and right-click. Select Maximize 3D Extents from the context menu to ensure that the grid or level is shown in every view in which it intersects.

SCOPE BOXES

Scope boxes are another way to control the display of grids and other datum elements in your project. Say you have a three-story steel structure sitting on a one-story concrete parking level, and suppose that the parking level column grid layout is different than the one for the steel structure. By enclosing the concrete level in a 3D scope box, you limit the propagation of gridlines so that in your views of the steel structure you won't have a confusing overlay of two grid systems. You use the scope box to limit their appearance in views. To add a scope box, go to a plan view and do the following:

1. Click and draw a plan view region that you want to control with the scope box.
2. Highlight the box and use the blue shape handles to further refine its horizontal extents.
3. For vertical adjustments to the scope box, go to an elevation, section, or 3D view and adjust the box using the handles.
4. To override the scope box in a particular plane, click the Element Properties button on the Options bar while the box is highlighted, which will display its element properties.
5. Click Edit in the Views Visible parameter, which displays the Scope Box Views Visible dialog box (see Figure 9.10).

6. In the Override column, change the value from Visible to Invisible (or vice versa, depending on what you require).
7. Close the dialog box.

To enable the scope box in a view, access the properties of the view and assign a scope box that you have created by selecting an existing scope box in that parameter field. Note that enabling the scope box will also limit the visibility of the entire view to that area, and that the view crop region will become locked to it.

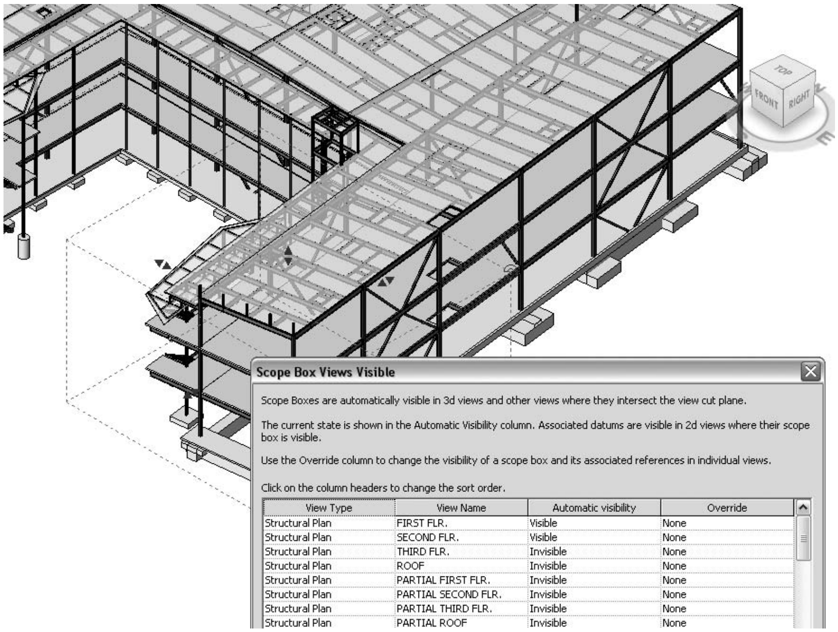


FIGURE 9.10
Scope boxes control the display of datum elements in your views.

Reference Planes

Reference planes have numerous uses. They establish two-dimensional planes in the three-dimensional space of your virtual environment, to which you can then attach your model elements. If you move the reference plane, locked objects will follow. They are also a vital part of the parametric family creation process (see Figure 9.11). For more on this subject, refer to Chapters 18 and 19.

In your model you can create reference planes and then set them as your current work plane. If you are working at an odd angle, you can work on elements on that plane while still in a straight plan view. Your elements can then be built in a basic two-dimensional approach such as the way roof framing was attached to a sloping diaphragm or reference plane in Chapter 7. Even though you were drawing the members in a plan view, the objects are still drawn on the sloping reference plane.

Once you have created a reference plane, you should give it a name for easy use and for later identification in the project.

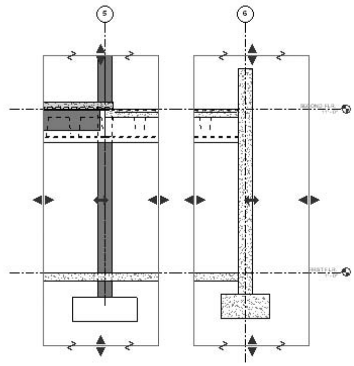


FIGURE 9.11
Locking pan joist beam family elements to reference planes

Annotation Elements

Now you are going to learn how to add annotation elements to your views. Not only are you creating a three-dimensional model in order to develop your design, but at the same time you are adding documentation so that your views will be ready to be added to your sheet set. Remember that annotation elements are view specific.

This section describes the addition of many types of tags that you will need for your documents in order to identify modeled elements. Text notes explaining details of the design must also be added, and as you will see, Revit Structure has it all covered. This section also explains symbols and the procedures necessary to load and add them.

Tags

Tags are element descriptors that are attached and display parameter values for that element (see Figure 9.12), such as a beam size or a footing mark. They should always be used, rather than writing text next to an object to describe it. Three options exist for tagging a modeled element. Of the three, Tag by Category is most commonly used. Tags can be found in the Annotations library installed with the software, with specific structural tags found in the Structural folder.

To add a tag to an element, do the following:

1. Click Tag ➢ Tag by Category on the Drafting tab (or one of the other two tag types).
2. The Options bar will display your various tag placement options (see Figure 9.13).
3. Designate Horizontal or Vertical placement of the tag.
4. Click the Tag tab on the Options bar to display a dialog box listing all the tags loaded in the current model and in all categories. You can select to make one current, or you can load others from the library directory.
5. On the Options bar, enable the automatic leader using the Leader check box. You can specify that the leader have its end attached or free, and you can set the leader length.
6. Hover over the object you want to tag until you see the appropriate tag. Revit Structure will attempt to match the right tag with the element.
7. Click on the screen to place the tag.

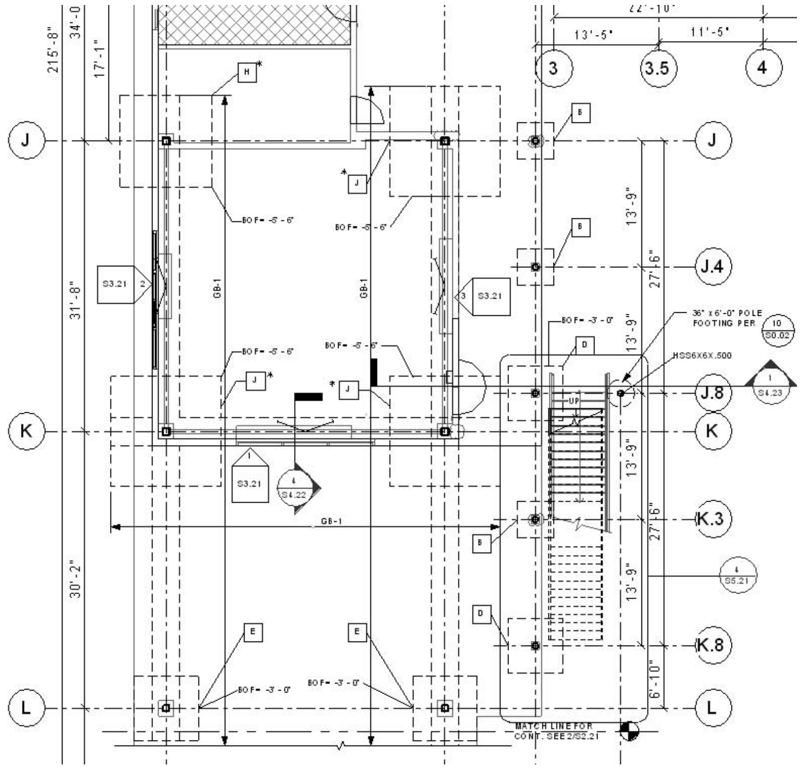


FIGURE 9.12
Typical annotated foundation plan

FIGURE 9.13
The Tags by Category options

When you move a structural framing member in your project, any attached tag automatically moves with it to maintain its original position in relation to that element. This will allow you to place a structural beam tag at the end of a beam, which then automatically moves and repositions itself as the beam moves.

Revit Structure ships with basic tags for all categories. You can also edit and create your own, which inevitably you will need to do in order to adapt the default tags to ones more in conformity with your company standards, and for special circumstances. For instance, the normal beam tag might read W12X26. But if you are working on an addition to an existing building, you might want to add an (E) in front of all existing framing to make it (E)W12X26. The wrong approach is to use the normal beam tag and then come back and add the (E) by using a text object in front of it. Instead, you can easily edit the tag and create a new type just for that purpose with the (E)

embedded in the tag. New to this version of Revit Structure, tags now support the combination of parameters and text in one label.

Here's the general procedure for editing the tag:

1. Open the tag family either by browsing to the structural annotation library, or by highlighting a tag that exists in your project and clicking the Edit Family button on the Options bar.
2. Once in the tag family, highlight the text label.
3. Click Edit Label on the Options bar.
4. Select new category parameters to add to the label (see Figure 9.14), or edit the existing label parameters.

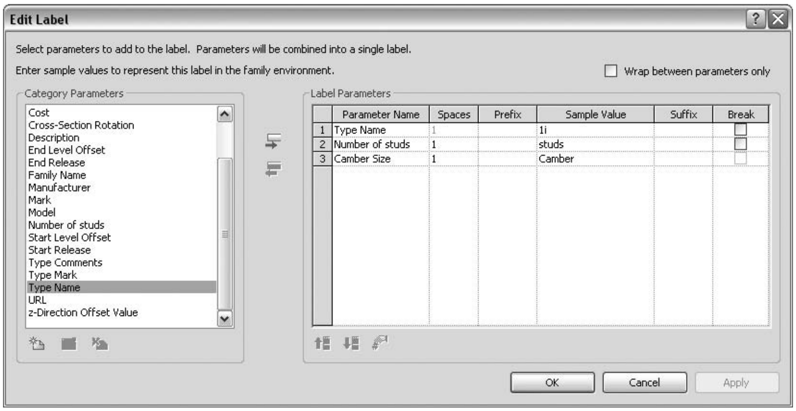


FIGURE 9.14
The Edit Label dialog box in the Wide Flange tag family

5. Edit the label by changing the sample value, or by adding prefix and suffix information.
6. Use the Wrap Between Parameters Only check box to create a multiline tag.
7. Use the Spaces field to increase or decrease the distance between the tag elements.
8. Once completed, save the view to a new tag family with an appropriate name, or if you're editing an existing tag, just save it.

These are the basics of how and why you add tags to elements in your model rather than adding a text note adjacent to an element. The next section shows you how to tag a whole plan at once.

TAG ALL NOT TAGGED

Use the Tag All Not Tagged feature to automatically add tags to multiple objects that do not already have tags. Sometimes you might find that you need to re-create a plan. At first you may think it a daunting task to reapply all the beam tags to the new plan. Maybe you think it is easier

to just edit the old plan. Well, with this option, tagging all the beams can be done in seconds, so don't worry about the time factor. To use this feature, follow these steps:

1. Select the elements you want to tag, or do not select any if you wish all to be tagged.
2. On the Drafting tab, click Drafting ➢ Tag All Not Tagged to display the Tag All Not Tagged dialog box.
3. Click the radio button to select all objects or the selected objects that you previously made.
4. In the Category section, select the type of object tag you wish to apply (see Figure 9.15).
5. Enable the Create Leader check box, and adjust the leader length if necessary.
6. Choose the orientation of the tag(s).
7. Click OK to create the tags.

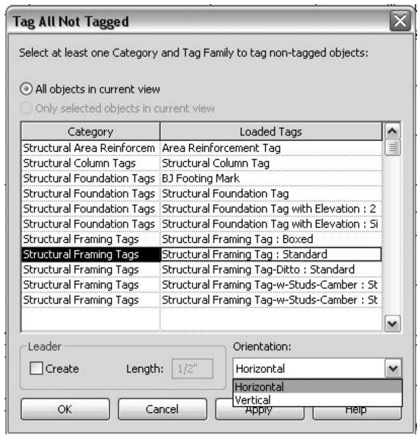


FIGURE 9.15
The Tag All Not Tagged dialog box

It's that easy, so give it a try. Even if some elements are tagged on the view, this function will find only those elements that have not been tagged and add tags to them. Revit Structure will not double-tag anything.

Beam System Tag

The Beam System tag (see Figure 9.16) covers a whole bay and indicates the common beam size and spacing of members in the framing bay. This tag is especially useful for wood framing in which members are closely spaced, typically at 16″ center to center and where tagging each individual member would totally clutter the plan. Simply click Beam System Tag on the Drafting tab of the Design bar, hover over the beam system, and click to place the tag.

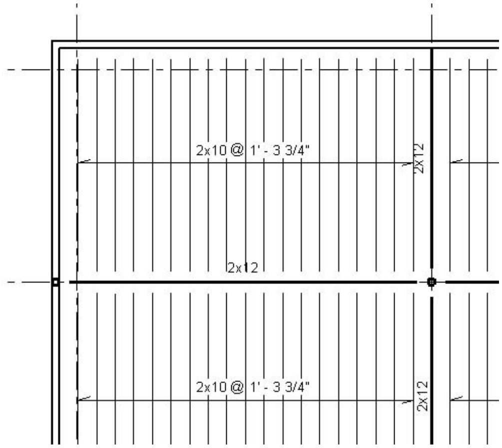


FIGURE 9.16
A Beam System tag applied to wood floor framing

SPAN DIRECTION TAG

The Span Direction tag (see Figure 9.17) is applied to structural floors as they are created, or it can be added later. The filled half arrow (see Figure 9.17) indicates the direction of the metal deck, and the open arrow represents the extent of the deck. The Span Direction indicator is important in details and sections because, once it is applied in plan, it will display the metal deck flutes correctly in section, making your sections much easier to generate. To change the direction of the deck, simply highlight and rotate the tag 90 degrees. Once the tag is placed properly, you can erase it and the deck flute orientation will not change. Unfortunately, the tag only works for floor objects and not for roofs. The deck profile cannot be added to a roof type and must be drawn manually within sections using a metal deck repeating detail.

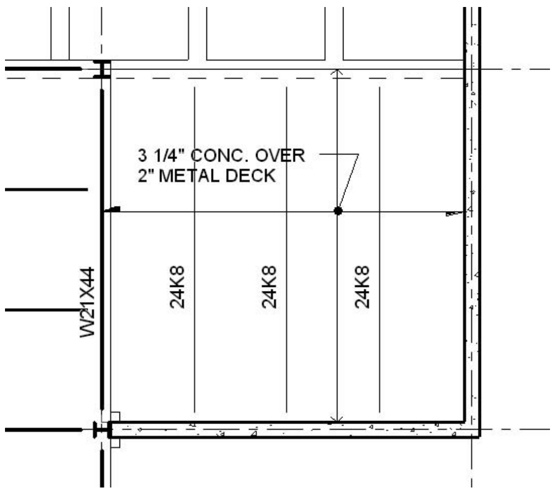


FIGURE 9.17
The Span Direction tag

Path and Area Reinforcement Tags

Path and area reinforcement tags (see Figure 9.18) are added to your concrete plans to indicate reinforcing patterns much like the Span Direction tag does for floors. For more information, refer to Chapter 10.

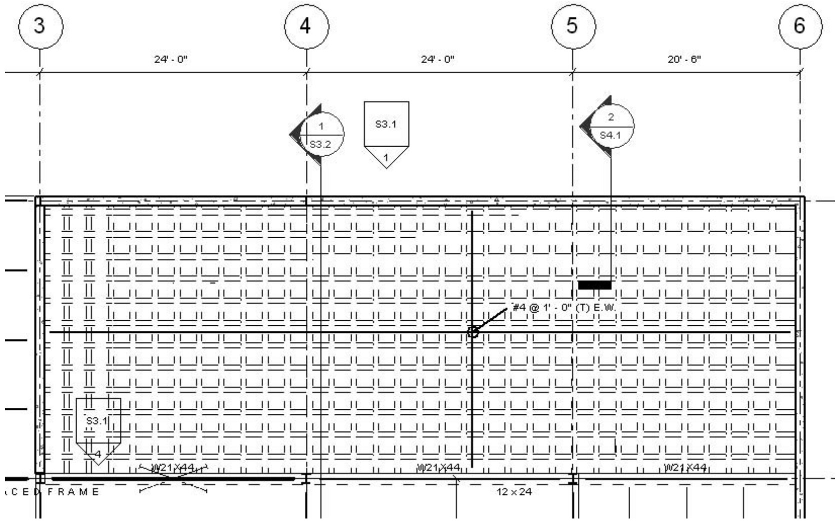


Figure 9.18
An example of area reinforcement in plan view

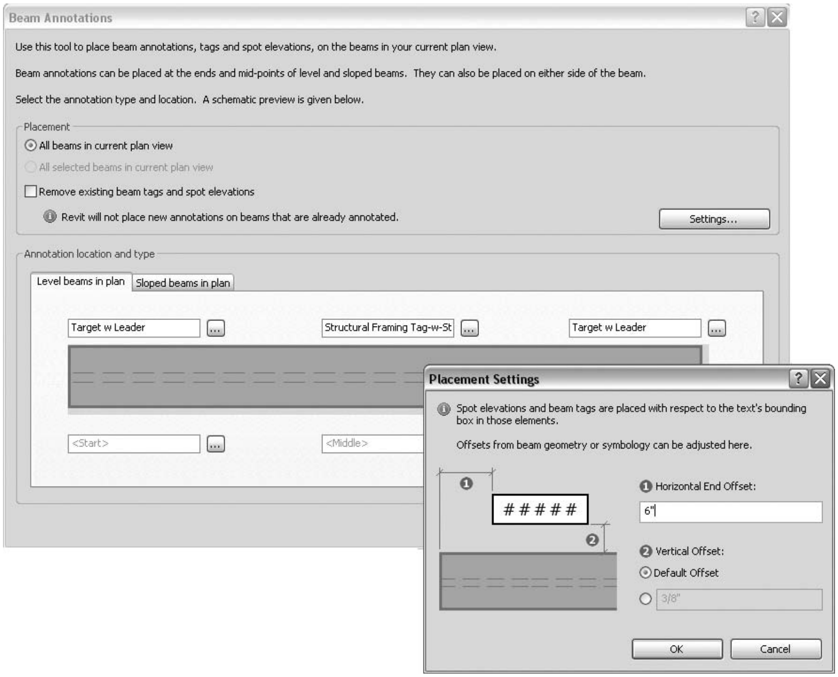
> ### Tagging vs. Dumb Text
>
> One of the most important concepts to grasp as you learn to use Revit Structure is that your modeled elements are tagged, rather than having text next to an object that has no connection to it, also known as *dumb* text. Tags are more intelligent than dumb text placed next to an element. When you attach a tag to an object, the tag displays a value, such as an isolated footing mark or a beam size, and connects directly to the underlying database of information. The value that displays represents one of the parameters of the graphical object, such as its type mark. Spread footings can be organized as a schedule on the typical details sheet, and will be sorted by their type mark. The same type mark value will display for the plan tag as well. The underlying database controls both of them. That is the parametric nature of Revit Structure, which helps it to be a powerful BIM solution: a robust, computable database that centralizes all information about the objects. Edits to the basic database are immediately reflected in all the views that the database controls. So avoid dumb text annotation whenever possible.

Beam Annotations

The Beam Annotations tool is an advanced tool for adding multiple tags at one time to a beam. Tags can be applied to all beams in a plan or to a selection set of beams. For instance, you can add framing tags to the middle of the beam, and you can add beam end elevations to the start

and end of members—all automatically. Figure 9.19 shows the Beam Annotations dialog box and its related Placement Settings dialog box. Each dialog box contains many placement options that you can set to meet your various conditions.

Figure 9.19
The Beam Annotations and Placement Settings dialog boxes



Exercise: Create a Structural Framing Tag

In this exercise, you will make a new tag for your project to adapt the basic Beam-Stud-Camber label. Your goal is to make the beam size tag display an (E) in front of it, the studs have brackets, and the camber display a c= in front of it, as shown here:

1. Start a new project.
2. Level 2 should be the default plan view. If not, double-click Level 2 to make it active.
3. Click Modelling ➢ Beam, then draw a beam on the plan.
4. Highlight the beam, and then click the Element Properties button on the Options bar.

5. Type a value of **24** for the Number of Studs value and **1″** for the Camber Size value.
6. Click OK to exit.
7. The stud and camber do not display because the tag does not contain labels for them. Select the tag.
8. In the Type Selector drop-down list, choose Structural Framing Tag-w-Studs-Camber : Standard to change the tag type so that these values will display.
9. With the tag still selected, click Edit Family.
10. Click Yes to open and edit the Structural Framing tag family.
11. With the tag family open, select the text and click the Edit Label button on the Options bar.
12. In the Label parameters of the Edit Label dialog box, add (E) as a prefix to the type name.
13. In the Number of Studs parameter, type **[** (left bracket) in the Prefix field and **]** (right bracket) in the Suffix field.
14. In the Camber field, type **c=** in the Prefix box.
15. In the Spaces field, increase the distance between elements to 3 for a better display.
16. Click OK to exit.
17. On the menu bar, click File ➢ Save As and change the tag's filename to **`Structural Framing Tag-w-Studs-Camber-existing`**. Place this file in your `Imperial Library\Annotations\Structural folder`. On the Design bar, click Load into Project.
18. Select the tag again, and use the Type Selector to specify the new tag type you just loaded.

Text

Not every element can be tagged in your project, so you must add text notes on various views (see Figure 9.20). As with other families, the text family is populated with many text style types formatted for your particular company needs and uses.

To place a text note in one of your views:

1. Click Text on the Basics, Drafting, or View tab on the Design bar, or choose Text from the Drafting menu.
2. Select the text style you want to use from the Type Selector (see Figure 9.21).
3. On the Options bar, assign the justification: left, centered, or right.
4. On the Options bar, select the leader type: No Arrow, One Segment, Two Segment, or Arc.
5. You can choose from two placement methods:
   A. Click once to place the top-left point of the text block in your view. This will make one long text box.
   B. Click the top-left corner and drag to pick the bottom-right corner. This will result in a specific box size with text wrapping. Remember to avoid adding hard returns or the text string will not wrap correctly when adjusted.

6. The Bold, Italic, and Underline formatting controls will then appear on the Options bar; use as needed.
7. Type the text string in the text box.
8. Click outside the text box to edit the text string.

To edit the text string in one of your views:

1. Select to highlight the text string.
2. Pick the move grip in the upper-left corner and drag in order to reposition the text object.
3. Click the rotate grip in the upper-right corner and drag to rotate the text object.
4. Click and drag the grips on the sides of the text box in order to resize the text box.
5. Click either of the two leader line grips and drag them to reposition the leader line.
6. To edit the text itself, simply click into the text box and change the text.

One important consideration for your text styles is whether the background of the text box will be transparent or opaque. An opaque text box will hide what is under it. You will need to decide which way you prefer to show your notes. Covering up what is beneath can lead to problems if it covers up something important. But sometimes that might be precisely what you want. There is a type parameter that controls this value, and you can adjust each text type as you see fit.

FIGURE 9.20
Adjust the text box using the grips.

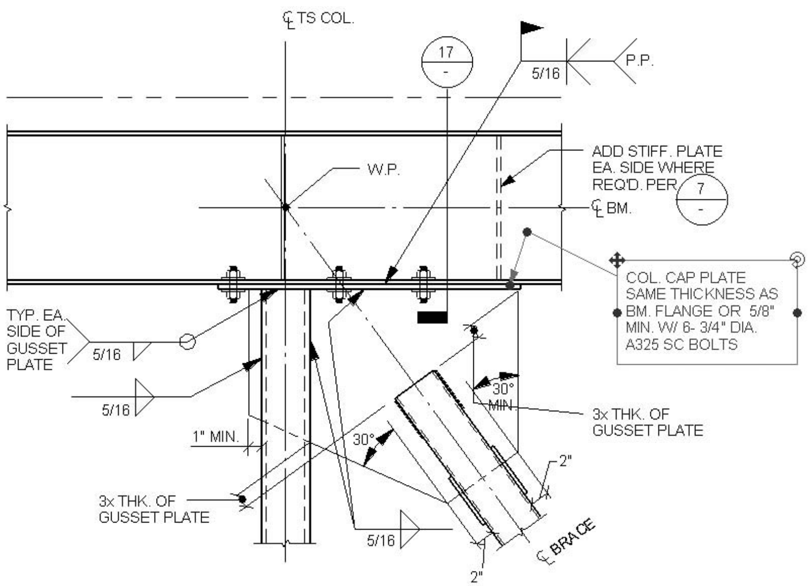


FIGURE 9.21
The Text Options bar

What about special characters in text strings, such as the centerline, plate symbol, or round symbol? They are important to structural annotation, and there is no text equivalent that can be easily used for the centerline or plate symbol. A centerline family is available, but it does not work in a text string, so you will need a workaround for that one. Many times, you will want an annotation that says something like "Centerline W18x," or "Plate 3/8" (see Figure 9.22) that includes the symbol along with some descriptive text next to it. The best thing to do is make your own annotation symbol.

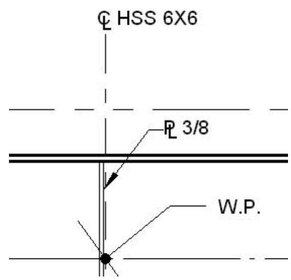


Figure 9.22
Simple centerline and plate symbol with added label

To make your own text annotation symbol for the plate text symbol:

1. Click File ➢ New ➢ Annotation Symbol.
2. Select Generic Annotation as the type.
3. Create the P and the L letters with transparent text and arrange them to make the symbol.
4. Create a label next to the symbol for the text addition.
5. Make the text string left justified.
6. Save and load the family into your project, and save it to your own annotation library.
7. To use the symbol, drag it from the Annotation area under Family in the Project Browser and click to place.

For some special characters, such as the round symbol, you can use the Windows Character Map, found in the Windows Start menu, under Accessories ➢ System Tools (see Figure 9.23). As you are typing the text string, you can insert special character symbols by doing the following:

1. Open the Character Map and browse through the symbols until you find the one you want.
2. Highlight the symbol and click the Select button.
3. Select other characters to copy to the string.
4. When you finish selecting, click Copy.
5. Switch back to the text string you are editing and paste in the characters (press Ctrl+V to paste).
6. Some characters have a Unicode value that can be seen at the bottom of the Character Map box, in which case you simply add that number in the text string.

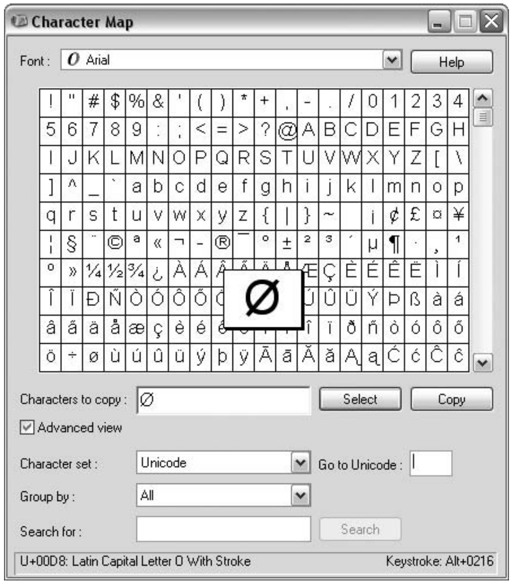


FIGURE 9.23
The Windows Character Map

Revit Structure handles text notes efficiently and effectively, as you have just learned, but sometimes you need to add large sets of notes. The next section will show you how to approach that task.

General Notes

General notes are text but require different attention than the notes we have been discussing. General notes usually take up a large part of your first sheet in a design set of documents, and they can be heavily formatted. They can be difficult to manipulate into rows and columns and to maintain on the sheet in the way you would like them displayed. It is easy to lose the formatting if you paste them in as an OLE object and edit them during the life of the project, although this approach can be used. If you use this import method, you can double-click into the OLE object in order to activate the original application and edit the notes. Until now, that was the best approach, but new tools are available that greatly reduce your effort.

If you are a subscription member, you will have access to the new Revit Structure Extension Text Generator, and you will undoubtedly want to start using it immediately. When you activate the command, a dialog box opens that allows you to insert and format your notes and configure how the notes will be displayed in columns and rows on the sheet (see Figure 9.24). Microsoft Word documents can be opened directly from the Extension Text Generator. This is just one of a whole set of excellent extensions that Revit Structure offers. If you've struggled with AutoCAD text for many years, you'll find this new Revit Structure function to be a breath of fresh air! Unfortunately for users of Revit Architecture, this extension is not available in that module yet.

The Extension Text Generator has a text import window where you can break notes into sections on the sheet, and a style area where you can create and edit the text format.

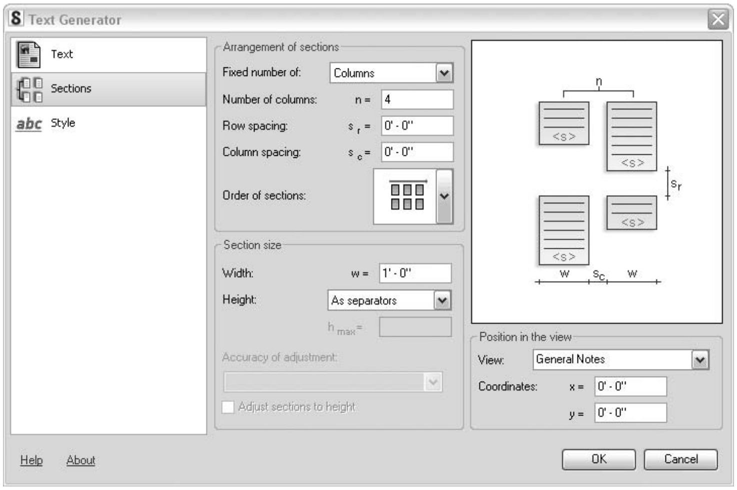


FIGURE 9.24
The Extension Text Generator allows easy general note formatting into rows and columns.

Here's the general procedure for using the text editor to import a DOC file (you must have Revit Structure Extensions 2009 loaded onto your computer):

1. Click Tools ➢ External Tools ➢ Extensions for Revit. This opens the Extensions Manager dialog box.
2. Double-click Text Generator in the Miscellaneous section to start the application.
3. In the upper-right area of the Text Generator dialog box, click the Select button and browse to the file you wish to import.
4. Select and import the file.
5. Add separators where you want to break the text lines into rows or columns.
6. Click Sections.
7. Enter the number of rows or columns into which you wish to arrange the notes.
8. Adjust the row, column spacing, and order of placement for each section.
9. Adjust the section width.
10. For the height of sections, you have two options:
    - **A.** Use the separators that you added in the text part of the dialog box.
    - **B.** Set the value to Maximum. Then, you can set a maximum height for the section.
11. Add the drafting view name for the notes and add a position (or you can just drag the note into position after import).

12. Click the Style button to access and apply text styles to the notes.
13. When done, click and Revit Structure will generate the notes as a drafting view.
14. Drag the notes onto your sheet (see Figure 9.25).

You will most likely have to fine-tune the notes to fit precisely. To edit the notes, go to the drafting view and click the Extensions Modification toolbar.

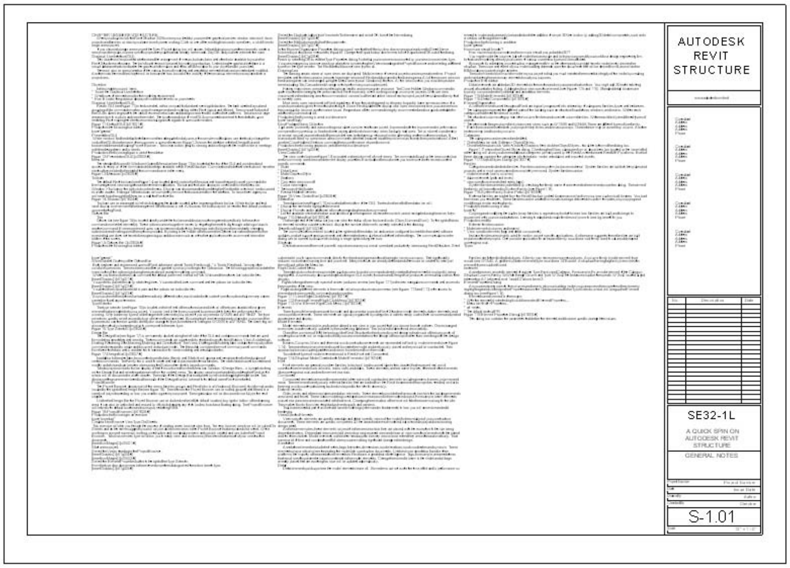


FIGURE 9.25 The sheet of general notes

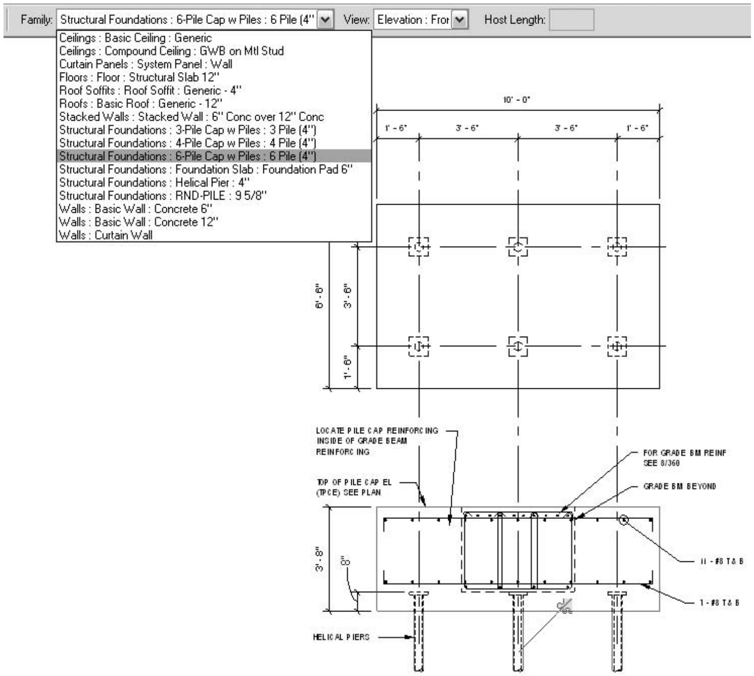
Keynotes

Keynotes are a system of defined notes that are applied to elements in your project and then displayed in a schedule or list on the side of the sheet. This can be an effective way to annotate your details in that you can apply keynote tags to objects to display the element type, the material type, or references to a keynote in the keynote schedule. For a complete description of keynotes, refer to Chapter 11.

Symbols

Symbols are used to provide a graphic representation of an annotation element in your project. As with other families, you can edit or make your own new symbols. It is important to build and manage a good symbol library as you transition into the Revit Structure environment. Moment frame connections (solid triangles) are an example of symbols, as are weld symbols and section marks. Figure 9.26 shows pile cap information in a legend.

Figure 9.26
A pile cap used as a legend component



Component symbols are especially useful when you're preparing a legend (see Figure 9.27). When you click Legend Component on the Drafting tab, the Options bar displays a pull-down list with all loaded components, which you can then drag into your legend view. You can configure the symbols to display in any detail mode, and either in plan or elevation view.

Creating a component and symbol legend (see Figure 9.27) is an important task that most projects require. You can easily assemble a legend view by dragging the symbol and legend components into it, as well as detail and model groups. Legend views also have the unique capability that they can be dragged onto multiple sheets. No other views can do that. To create a legend for your project, do the following:

1. Create the initial legend view by clicking View ➢ New ➢ Legend.
2. Give the legend a name and scale.
3. On the Drafting tab, click Symbols and drag the symbols you want on the legend into the view.
4. Click Legend Component and select an item from the Component drop-down list on the Options bar.
5. On the Options bar, choose the Plan or Elevation view for the item.
6. Place the component in the view.
7. Add any other detail or group components to the view.

8. Create descriptive text next to each item.

9. Drag the legend view onto a sheet.

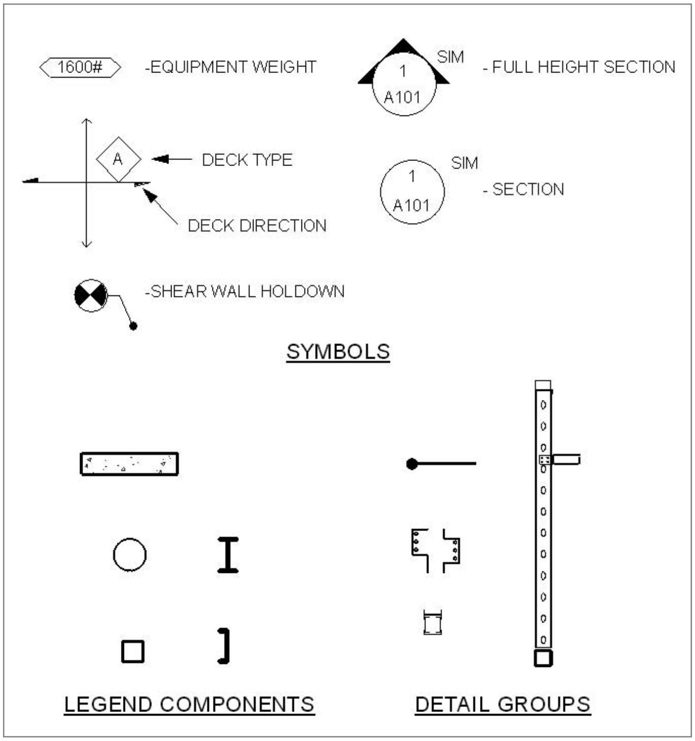


FIGURE 9.27
A legend view displaying various components

WELD SYMBOLS

Weld symbols are loaded in from the Structural annotation library. You access them in the family area on the Project Browser or by choosing the Symbol command on the Drafting tab of the Design bar. Unfortunately, the Revit Structure weld symbol that ships with the product is a bit clumsy and awkward to use. The symbols themselves are too big. And for some strange reason, there are 0″ values inserted for all the fields that must be cleared every time you use the symbol. You will probably find yourself trying to edit the family when you start working with it, or you will be looking on the Internet for better families that might have been shared. What is needed is a graphical dialog box that allows you to fully and easily configure the symbol before you place it. It is a bit astonishing that the developers have not found the time to improve such an important symbol for all who are documenting a steel structure.

To use the default symbol, follow these steps:

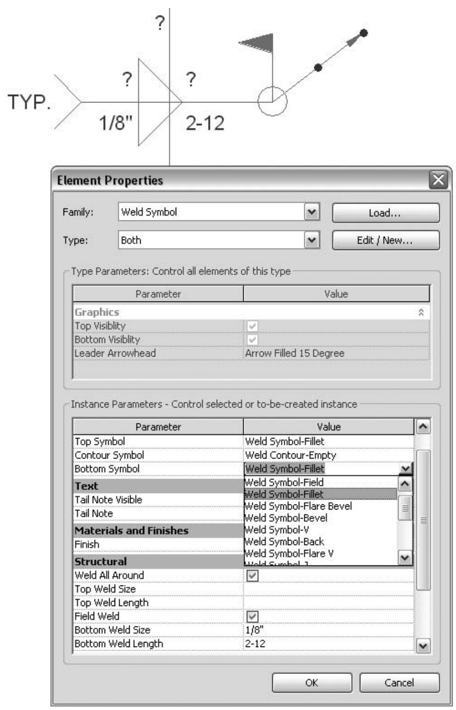
1. Choose Weld System Both, Bottom, or Top from the Family Annotation area in the Project Browser.

2. Drag and place the weld symbol into the view.

2. Highlight the weld symbol and click the Element Properties button on the Options bar.
3. In the instance parameters, edit the Top and Bottom symbols as required (see Figure 9.28).
4. Add tail notes, finishes, weld sizes, the field weld symbol, the all around symbol, and others as required for your detail.
5. Use the Symbol Left checkbox to orient the leader line to the left or right of the symbol.
6. Click OK when you are done.

Many of the values can also be added or edited by highlighting the weld symbol after placement and clicking into the text fields.

FIGURE 9.28
The Element Properties dialog box for weld symbols



That concludes our section on annotation. A well-annotated set of documents is vitally important in project preparation. As you have seen, Revit Structure offers lots of help for you in this area as you are creating your documents.

Now let's move on to adding 2D detail elements to your various views in order to show those elements you did not want to or could not model.

Detail Elements

Drafting views have no model elements in them. They are strictly two-dimensional views where you use traditional CAD tools to draft a detail. Section, callout, and elevation views, on the other hand, contain elements from the model as well as additional 2D detail elements. Detailing your project requires working with both types.

If you or your company have a subscription contract with Autodesk for Revit Structure, there is an excellent command for detailing in the Revit Structure Extensions for 2009: the Freeze Drawing extension. Any section or elevation with model elements can be turned into a drafting view. All model lines are copied and used as detail lines in a new drafting view. If you are worried about shifting model elements, or wish to make a typical detail, this extension will fit the bill.

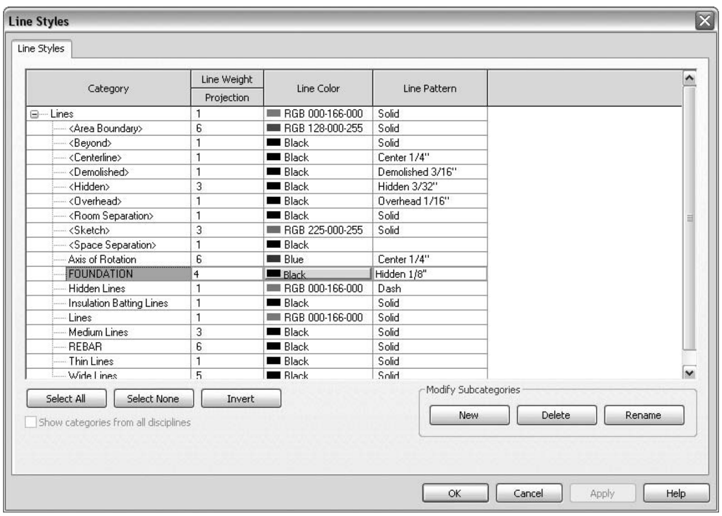
Detail elements help you complete your sections and details. Those that you decide are not necessary to model can be added as 2D elements to your detail views. For instance, you will model the steel column and the spread footing in your project, but you may want to show the base plate, bolts, and grout as a 2D detail group of components in an enlarged section. That is probably the case if you are working for a structural design firm where you are interested in showing only the various typical types of elements. If you are working in a structural detailing firm, on the other hand, you may choose to model all the connection elements so that they can be detailed and scheduled into piece lists.

The following sections introduce you to the various tools that you will use time and again to complete your details.

Detail Lines

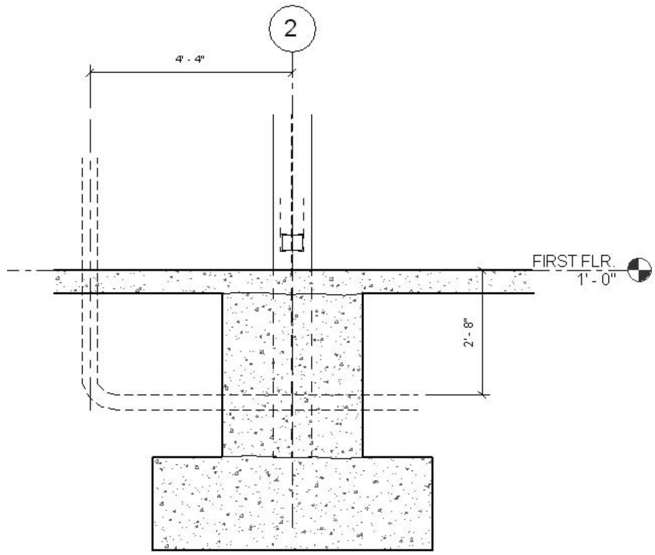
Detail lines are the basic line drawing tools that you will use for 2D detailing. Use them to finish drafting views that are cut from the model and 2D drafting views as well. Click Settings ➢ Line Styles to access a dialog box that enables you to edit existing and create new line types (see Figure 9.29). These line styles are used not only for detailing lines but also for the Linework tool. With the Linework tool, you can change the line style of most model lines if they are not displaying properly.

FIGURE 9.29
Line Styles dialog box



Detail lines can add important information that you might not want to model, such as piping through footings (see Figure 9.30).

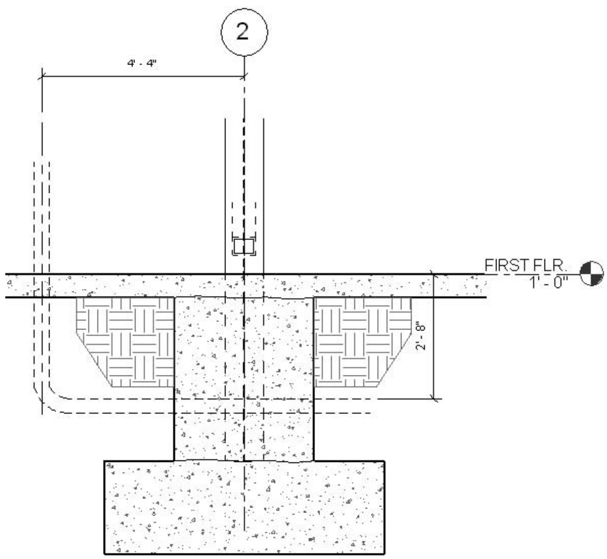
FIGURE 9.30
Detail lines show piping run through a slab and grade beam.



Filled Regions

Filled regions provide hatching patterns that you use for detailing purposes (see Figure 9.31). The patterns must be enclosed in a continuous boundary of lines. The boundary element can be invisible, or each segment can be visible and use a different line style.

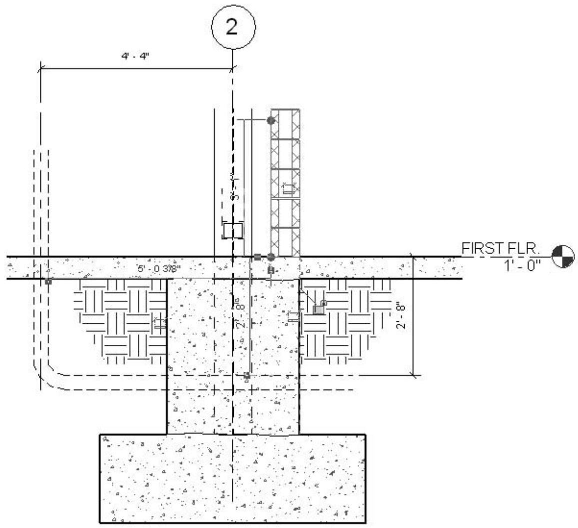
FIGURE 9.31
Earth hatching added to detail



Repeating Details

Repeating details are useful in a detail view when you're adding repetitive 2D elements such as punched steel studs, 2D concrete reinforcing in section, metal deck flutes in section for a roof, or masonry block wall coursing (see Figure 9.32). Detail components can be easily spaced at intervals of your choosing, and applied by specifying start and end locations.

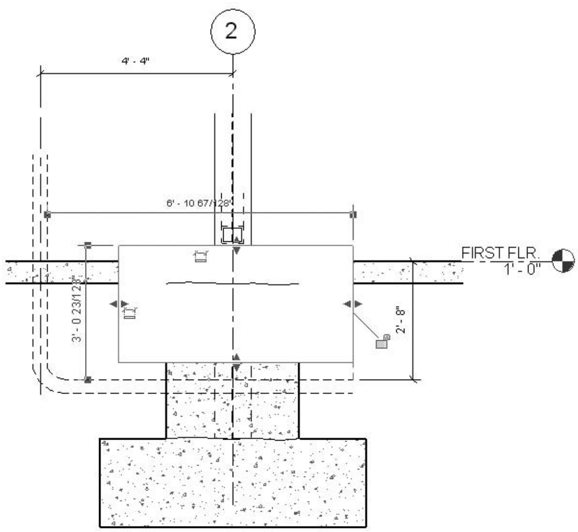
FIGURE 9.32 Repeating block detail



Masking Regions

Many times in your struggle to make model elements in a view display correctly, the best practice may be to cover a problem area over and to finish in 2D. That is where the masking function becomes useful (see Figure 9.33).

FIGURE 9.33 Masking region covering a portion of the detail



Detail Components and Groups

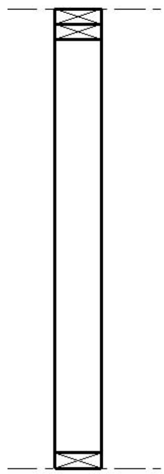
Detail components are 2D representations of model elements, such as wide flange beams in section, that can be used in detailing views. Detail groups are assemblies of detail components that you want to use more than once. You will be able to drag them into your views from the bottom of the Project Browser for easy insertion.

Embedding Detail Components in Modeling Families

When you are cutting a section on your model, you want the section to be as complete as possible without adding lots of detached lines and detail components. When the model elements move during the design process, the detail components usually do not move with them. Therefore, it can be quite time consuming to continue editing and relocating these elements. One way to avoid this is to build the detail components right into your model family. That way, they are automatically added and can be easily displayed in your section work and they will move with the model element in which they are embedded.

A good example of this approach is the creation of a standard 2 × 6 wood-bearing wall type. In a section view you add the top and bottom wood plates to the wall representation (see Figure 9.34) by creating a profile for the flat 2 × 6. Within that profile, you add a 2 × 6 detail component. That profile is then added to the wall type.

Figure 9.34
Wood wall type with added profiles



The following general procedure will lead you through this process. First you create the required profile for the 2 × 6 flat stud plate:

1. Start a new project.
2. On the menu bar, click File ➤ New ➤ Family and select Profile from the list of available template types. Then click Open.

3. On the menu bar in the profile file, click File ➢ Load from Library ➢ Load Family.
4. Load in the component. In the Imperial Library, select Detail Components, then Div 06-Wood and Plastic.
5. Select 061100 – Wood Framing, then choose Nominal Cut Lumber – Section for the family.
6. Click Open and then specify 2 × 6. Click OK.
7. On the Design bar, click Detail Component and place the 2 × 6 near the origin.
8. On the Design bar, click Modify.
9. Select the 2 × 6 and rotate it 90°, then center the 2 × 6 on the vertical reference plane. Move the bottom line of the 2 × 6 up to the horizontal reference plane.
10. On the Design bar, click Lines, select the Lock check box, and then draw a rectangle over the 2 × 6 exterior lines.
11. Highlight the detail component. Click Visibility on the Options bar. Deselect Course and click OK.
12. The profile is now complete. On the menu bar, click File ➢ Save and save the file as **Wood – 2x6 profile**.
13. On the Design bar, click Load into Project.

Now you add the profile with the embedded component to the wall family definition:

1. On the Modelling tab of the Design bar, click Structural Wall.
2. Click the Element Properties button on the Options bar.
3. Click Edit/New, then change the wall type to Generic 6″.
4. Click the Edit button in the Structure value field.
5. In the Edit Assembly dialog box, click the Preview button.
6. Select View: Section: Modify type attributes in order to get a side view of the wall.
7. Click Sweeps, then click Add to insert a new profile (see Figure 9.35).
8. In the Profile field, click and select the Wood 2 × 6 profile.
9. In the Offset field, change the value to –3″.
10. Click OK until you exit all the dialog boxes.

Now draw a portion of wall in plan view. Create a section view through the wall. When you go to that section view you will not see the flat stud plate at the bottom of the wall. That's because the view is in Coarse detail level and the Visibility option for that mode was turned off in the profile family. On the View Control bar, change the detail level of the section to Medium and you will immediately see that the detail component appears at the bottom of the wall. For practice go back and using the same procedure add the two top plates to the wall definition.

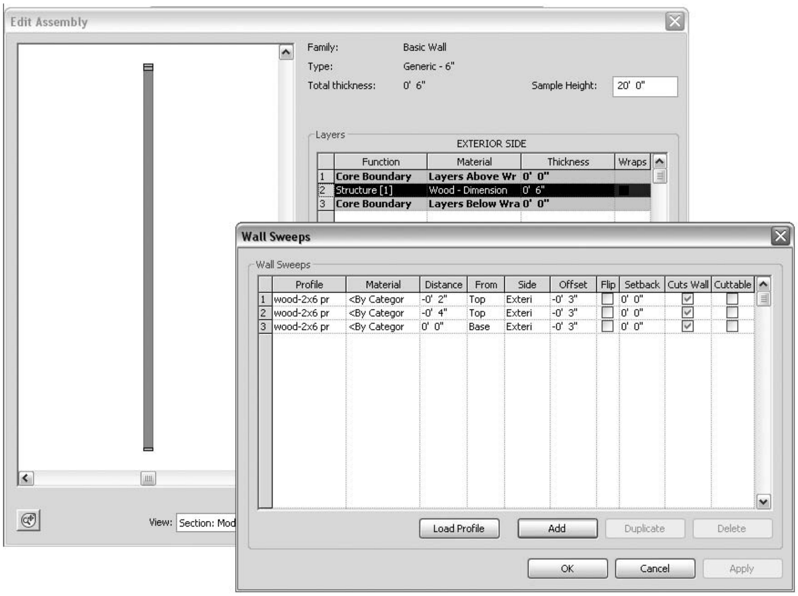


FIGURE 9.35
Adding the profiles to the wall type

Using embedded components is the best practice for detailing. Now if the wall moves, the embedded detail components go with it. A further refinement on this procedure would be to create a profile that uses the flat 2×6 stud we created and also adds a bolt detail component. That profile would then be specified at the bottom of the wall to indicate the connection of the wall to the floor concrete. Chapter 19 takes this wall family even further by creating a full wood shear wall that displays well in both plan view and section view without a lot of added 2D work to complete it.

Now that you have an idea of how to use Detail Elements let's explore how you will create and manage your typical details.

Typical Details Sheets: Creating and Managing

Typical detail preparation is a vital task in documenting any building project, and there are several ways to approach it in Revit Structure. One way is to capture sections from the model that you can then generalize to cover many typical conditions, such as typical foundation conditions (see Figure 9.36). Rather than relying on a library of sheets of foundation details, you selectively prepare what actual conditions you encounter, using the model as your guide.

Undoubtedly, you will have a standard set of details in AutoCAD or other CAD program that you will want to create eventually in Revit Structure. Most of these details have no model connection and are inserted as 2D drafting views into your project.

To avoid becoming overwhelmed with transforming your whole workflow to modeling at once, you may want to keep your typical details in your CAD system for your first number of projects. That will give you a blended construction document set of Revit Structure and CAD sheets. The CAD sheets can be managed and printed solely from the CAD program. As your staff is transferring skills from CAD to BIM, you can still keep both systems going and keep everyone productive. You can also simply build the details from scratch in Revit Structure.

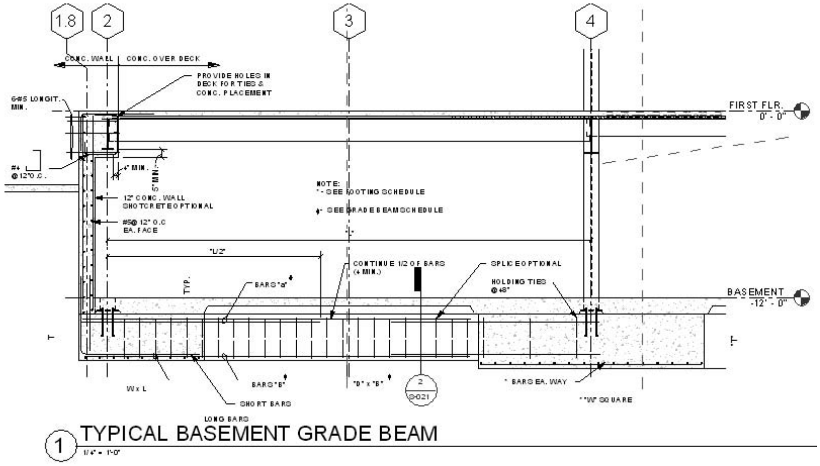


FIGURE 9.36 Modeled foundation section generalized as typical

If you decide to have a mixed set of documents, a good practice is to link the CAD detail sheets into your Revit Structure project but still do all the work in CAD. This approach allows all the printing for the project to be performed in Revit Structure. That is an important consideration because it can be difficult to coordinate one project in both formats and have them look the same when it comes time to print. At the very least, you will have to maintain two title block sheets, one in Revit Structure and one in CAD.

Revit Structure has several commands that allow you to import and export drafting views and to create libraries of standard details (see Figure 9.37). It works quite well. You can import or export single drafting views or entire sheets of drafting views. Note however that if the sheet has any views with mode elements in them, none of the views will be exported. Next you will see how to use these commands.

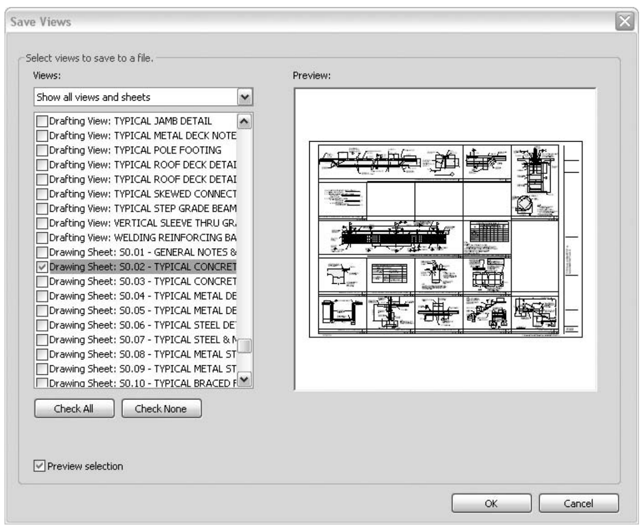


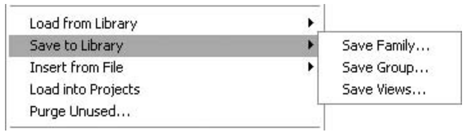
FIGURE 9.37 Saving sheet of views

Save to Library

You can save families, groups, and views from your project (see Figure 9.38) with this command. For instance, if you want to save views to a typical details library, do the following:

1. On the menu bar, click File ➢ Save to Library ➢ Save Views.
2. In the Save Views dialog box, select by checking the box next to its name the sheet or view names you want to export.
3. In the Save As dialog box, browse to the folder where you want to store the views.
4. Click Save.

FIGURE 9.38
Save to Library options



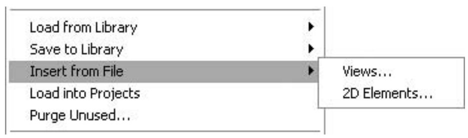
If you just choose the sheet view, it will be created as a file with all the drafting views included.

Insert from File

Views exported from other Revit Structure files, or whole sheets of views, can be inserted with the Insert from File command. The procedure to insert drafting views is as follows:

1. Click Files ➢ Insert from File ➢ Views (see Figure 9.39).
2. Browse to the folder and select the RVT file where the view(s) reside that you want to insert, or simply select the sheet you want to insert.
3. Click OK.

FIGURE 9.39
Insert from File options



You can insert 2D elements from a saved model view by using the 2D Elements option. This option allows you to copy 2D detail components such as repeating details and filled regions from a view that contains model elements.

Real World Scenario

Importing and Linking Details from AutoCAD

One of the biggest problems in setting up Revit Structure for first-time users is establishing proper import line weight settings. If you use American Institute of Architects (AIA) layering standards in AutoCAD, it is easier, but if they have all kinds of tweaked layers—and of course third-party SHX fonts—they are going to be shut off in terms of migration. Some firms have interns importing AutoCAD details into Revit Structure as they build their detail libraries since it can be a time consuming and repetitive task. The template they use has to be right on, or those kids will just be creating much more work for themselves massaging the detail once it gets imported. It is important to have all of the filled regions premade so that when the AutoCAD hatch is properly transferred to a Revit Structure pattern upon import, they will transfer to the predefined regions. Line types should be created so they transfer the AutoCAD lines directly without further editing.

Imported AutoCAD layers with no mapping to a Revit Structure line style will create a new line style with the AutoCAD layer name. AutoCAD hatch patterns and text styles will also create long unwieldy names when inserted. This can be problematic, and your details can easily be polluted with this type of junk. Some firms create AutoLISP routines or scripts to alter the detail in AutoCAD to help avoid that type of mess when the detail is then imported into Revit Structure.

Importing AutoCAD hatch patterns must also be approached carefully. A detail imported from AutoCAD will insert as a block. When you highlight the block, you have the option to explode it partially or totally. If you explode it totally, the hatch pattern will be exploded into its little line segments. This can greatly affect the performance of your file and should be avoided. Instead, do a partial explode. In that case, the hatch patterns will not be exploded. You can then highlight them and change the pattern to a named Revit Structure pattern.

That ends our discussion on documenting your model. It is a huge subject that could fill a whole book. With this introduction of some of the most vital factors of documentation you now should have some direction on how to prepare your projects. Remember that documentation is half the battle and must be carefully planned and executed in order to maximize the use of your model.

The Bottom Line

Add datum elements to your detail and section views. Datum elements are necessary for your model because they are the anchors for your objects. Grids, dimensions, spot dimensions, and reference planes are basic constraints for elements within the model, and give it the ability to flex as you are working through changes in the design of the structure.

Master It Datum elements form the basic constraints for your project. Using Notepad, write a paragraph explaining how Revit Structure constrains the various modeling elements.

Add annotation elements such as text, tags, and symbols. Once the model is moving forward in development, you need to efficiently add identifying tags, beam annotations, and text to your various views in order to document your design and prepare your sheets. Tagging elments is an essential task since it taps into the properties of the object. If the object changes type, the tag automatically updates. That then allows you to use the model as a physical database for building schedules of many kinds. Text and symbols also are used to further the documentation of your model.

Master It Open `Dataset_0901_Begin.rvt` (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`), then go to the second floor plan. On the second floor, load and tag all steel members. Add a Beam System tag to at least one bay. Add a Span Direction tag to the floor. Go to the first floor and tag the columns. After placing the tags, highlight and use the grips to align them with one another for a better display. Add grid dimensions.

Add detailing elements such as detailing lines and filled regions. Not everything is modeled. It takes experience to find the correct level of modeling in your project. For instance, columns are modeled but base plates are not in a typical American design firm. But when taking sections and creating details, you have to add that information in 2D over the modeled objects. So you add detailing lines to show the column base plate and perhaps some earth hatching around it. These are detailing elements.

Master It Open `Dataset_0902_Begin.rvt`, then go to the callout of Section 6. Add detail lines to show piping 4′-0″ to the left of the column going through the slab, turning 90 degrees and going through the slab. Use a hidden line style. Add earth hatching below the slab using a filled region. Add a repeating CMU component wall to the right of the column with its outside flush with the grade beam below.

Create a typical details library. A critical task to accomplish if you want your project to be totally documented in Revit Structure is the management of typical detail libraries. Typical details can be imported from your 2D CAD library or created from scratch in Revit Structure. You import Revit Structure details individually as drafting views, which are then added to sheets. They can also be inserted as part of a whole sheet. In similar fashion, you can export individual drafting views or sheets of drafting views to use in another job or to add to your Revit Structure library of details.

Master It You have a new project to start and want to transfer your model and drafting views from an already completed project. How will you transfer the drafting views to the new project? What is the best way to transfer a section with model elements in it to another project as a typical drafting view?

Chapter 10

Modeling Rebar

The power of Revit Structure is no more evident in any capacity than in the procedures involved in the placement of reinforcement. To say this topic ties it all together is an understatement. In flat AutoCAD, AutoCAD Architecture, and Microstation, reinforcement is simply a separate entity. You draft it yourself. It stands to reason that an application that allows you to place, for example, a #4 bar with an on-center spacing as opposed to a "donut," is a much better way to think when it comes to drafting.

In Revit Structure, reinforcement is supplemental to the item that hosts it. With that being said, Revit Structure gives you the ability to control the performance of reinforcement long before you even place a reinforcement bar (rebar) in a wall, a footing, a column, or a pier. You are provided with settings that allow you to control reinforcement cover in both the object's element properties as well as global Revit Structure settings. Revit Structure allows you to place predefined bars via a powerful new tool called the Shape Browser, and also gives you the ability to create your own rebar and add it to the library without leaving the model to create a new family.

In this chapter you will learn to:

- Draft a 2D rebar
- Configure rebar settings
- Draft a 3D rebar
- Add rebar shapes

Drafting a 2D Rebar

There are many items in Revit Structure that should be fully modeled in 3D. There are also just as many items that, simply put, should not. Concrete reinforcement is probably the one item that falls into that gray area of "Should I model all of this in 3D, or do I draw it in?" It can be a tough decision to make in terms of what you are looking to get out of the project. On one hand, if you do model the entire project with the proper reinforcement, you can rely on the quantity take-offs and the reinforcement scheduling to be accurate. On the other hand, if you model all of the reinforcement, it will take longer, and the model will quickly grow in size. We recommend that you look at the overall project. If it is a 650´-long military barracks, or a football stadium, then you might want to consider placing 3D reinforcement only to the items you will be specifically detailing. If it is a more manageable project in terms of size, then the advantage of fully reinforcing the model will outweigh the disadvantage of the initial time spent and the subsequent increase to the file size.

Sometimes, this decision is not easily made. There will be times where you do need to illustrate reinforcement, but cannot afford either the time or the file size. It is here where you are going to want to simply *draft* the reinforcement. Before you learn how to perform this task, note that drafting can be done in any view. There is a misconception that drafting has to occur in a section or a drafting view. Although we *will* be drafting in a section, we are also going to be drafting in a plan. But before you start throwing lines at the model, there are a few settings you may wish to configure beforehand.

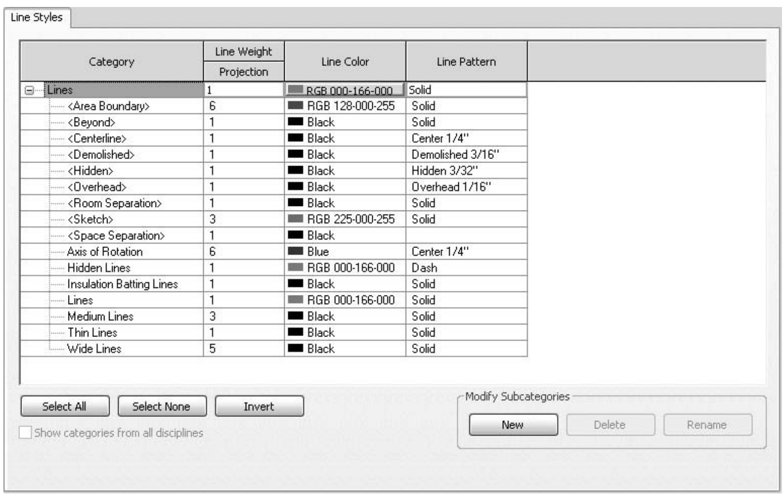
Line Styles

Since this chapter is going to discuss drafting reinforcement, it would be prudent to look at line styles. Line styles and how they pertain to your company standards are covered in depth in Chapter 17, but let's take a glance now. If you are going to be simply drafting reinforcement as you did in AutoCAD, it makes sense to have a template set up that allows you to choose the line types you want. Revit Structure does have some suggestions, but most firms need more.

Choosing Settings ➤ Line Styles opens the Line Styles dialog box. By default, only one category, Lines, is available, as shown in Figure 10.1, and it is not expanded. If you expand the Lines category, the rest of the lines will be exposed. Notice the top few choices are bracketed. This allows you to physically overwrite a line defining a Revit object using the Line Work tool. What you are after are the bottom three lines: Medium, Thin, and Wide Lines.

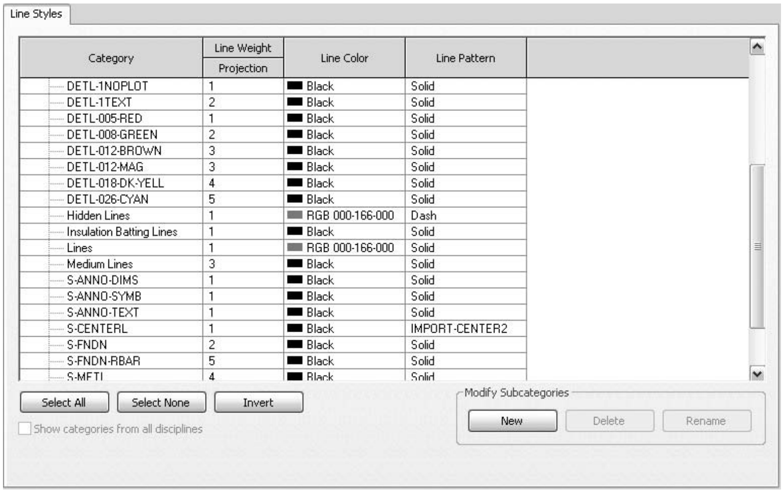
At this point, it is worthy to note that, while you can add whatever lines you wish, you should look at what you are provided with here. Since many readers have been using AutoCAD for a long time, it is apparent that the industry standard is to have a layering system that often exceeds 200 layers. Sometimes the system is color dependent, and sometimes not. In Revit Structure, however, you have a chance to change that. Medium, Thin, and Wide—how much more do you really need for the illustration of your model? Line work in Revit Structure is supplemental to the overall scheme of things. If you need to display items in a different manner, you can simply configure the Visibility/Graphic Overrides settings for that view.

FIGURE 10.1
The Line Styles dialog box lets you add and configure existing and additional line work.



Notice the Line Color column. This color will *plot* as the line color illustrated in this dialog box. It may be a good idea to change this color to black (if you are used to AutoCAD). With Revit Structure, sometimes you need to let go of the past in terms of seeing colors all over the drawing, but only as long as letting go of the past is for the better. Avoiding 200-plus layers and colors *is* for the better. Given that, there are going to be some new lines introduced. Only having three line styles may be pushing it in terms of simplicity. The dialog box in Figure 10.2 illustrates line styles that are translated from an AutoCAD drawing.

FIGURE 10.2
You can add additional line styles, such as S-ANNO-DIMS, similar to AutoCAD.



To make a new line style, simply click the New button in the Modify Subcategories field in the lower-right corner of the dialog. You can then rename the new line style and assign it a line weight.

Now that the line styles are set, it is time to start drafting a rebar in the model. Remember, as mentioned earlier, you can draft in any view, including a plan view. To practice configuring line styles, follow these steps:

1. Choose Settings ➢ Line Styles.
2. In the Line Styles dialog box, set the Weight, Color, and Pattern options as you wish.
3. If you want, you can click the New button at the bottom of the dialog box to add a new pattern.

With the line types in place, the process for single line drafting will be much easier and similar to the method you have grown accustomed to using over the years.

Drafting in the Model

To provide drafting lines to a model, you use the Drafting tab on the Design bar. When in a view such as a floor plan, navigate to the Drafting tab on the Design bar as shown in Figure 10.3. On this tab, you can then click the Drafting Lines button.

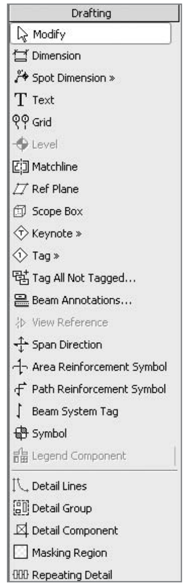


FIGURE 10.3
The Drafting tab on the Design bar

Once you've selected the Detail Lines command, you can proceed to draw the rebar. In the Options bar, you will see the Type Selector is populated as shown in Figure 10.4. This will give you access to the styles you just reviewed in the Line Styles dialog box. Select the appropriate line type, and draw in the reinforcement. It's really that simple. And to be honest, it's really just like AutoCAD! You start the Detail Line command, assign the correct line style (also known as a layer), and draw as needed.

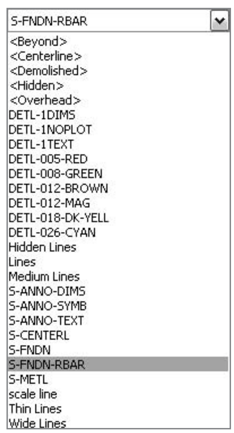


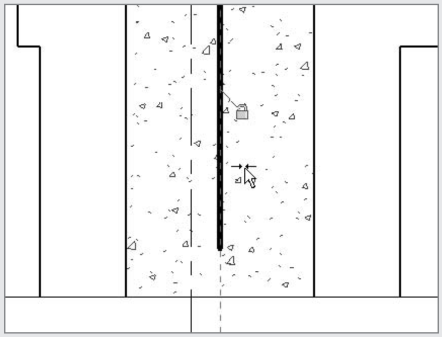
FIGURE 10.4
The Type Selector drop-down on the Options bar will look like this.

Since you are simply drafting, you are not restricted to planes or boundaries. This means that as you draft, you can place the lines anywhere you want in the view.

Since the rebar can be sketched freely, items like foundation pins can be more readily placed, as shown in Figure 10.5.

Lock It to the Model!

As you draft, remember that you can still align and lock the drafting lines to the objects you are drafting over. This will give you the advantage of leveraging at least some of the inherent Revit Structure functionality. When the wall moves, the line will move with it, as shown in the following illustration.



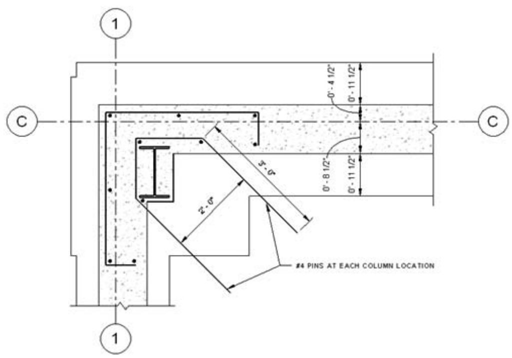


Figure 10.5
Detail lines and detail components to create a reinforcement plan detail

Adding reinforcement by using detailed lines is nice to be able to do, but does not take full advantage of the 3D modeling environment. Revit Structure will allow you to do this same task, but the reinforcement elements are truly part of the model. This means you can not only place the reinforcement, but also include it in schedules and quantity take-offs.

Rebar Detail Components

In AutoCAD it's called a donut. Some firms will make a block out of that donut, so if it changes, all of the bars will be updated. What are we talking about? We are referring to the perpendicular bars that need to be placed into the model. You do this by using the Detail Component command on the Drafting tab of the Design bar. Once you click this button, you can then go to the Type Selector and pick the Rebar Detail family. You can then place the vertical (horizontal in the case of a plan view) bars in the model, as shown in Figure 10.5. If you do not have the rebar family

loaded in your model, click the Load button on the Options bar. Browse to Detail Components, Structural, and then Concrete, `Reinf Bar Section.rfa`. Once this new family is loaded, you will see that the Type Selector has different numberings for the bar sizes. You now have a wide choice of bars. Who says you can't draft in Revit Structure?

Working with this "free to just draft" mind-set, let's look at something here. Suppose you still had the freedom to "just draft" but the ability to use intelligent items instead of detail lines? Before, in Revit Structure this procedure was somewhat of a chore. In Revit Structure 2009, a new approach to reinforcement a model has been developed. To draft reinforcement, use the following steps:

1. Open a plan or a section view.
2. Go to the Drafting tab on the Design bar.
3. Click on the Detail Lines command.
4. Set the appropriate line style in the Type Selector.
5. Draw as you always have done in the past!

So, as you can see there is a tremendous benefit to using the Revit Structure method of adding reinforcement with positive downstream effects.

Placing 3D Reinforcement

Placing a rebar in Revit Structure has just gotten a whole lot easier! The biggest upgrade in the software has been made in this area. The best thing about it is that you can still basically draft reinforcement as we explained earlier in this chapter, but now the reinforcement will allow you to specify centering and will also let you to choose from a menu of predefined shapes.

Of course, where there is 3D there are settings that will need to be looked at before the bars are actually "placed" in the model. We need to know how thick our rebar is going to plot, and we also need to see where the rebar will appear visually unobstructed. Some of these settings are located in different areas. The following sections will describe where to access these settings; we'll begin with the Object Styles dialog box.

Object Styles

The first item we'll examine is how the bars will appear in terms of their plotted line weight. The out-of-the-box appearance has foundation walls thicker than the actual reinforcement, as shown in Figure 10.6. This just can't be! Although you may have the settings correct at your firm, let's get a handle on where these settings are and how you adjust them.

Select Settings ➢ Object Styles, as shown in Figure 10.7.

In the Object Styles dialog box are three settings you need to configure:

- Structural Area Reinforcement
- Structural Path Reinforcement
- Structural Rebar

FIGURE 10.6
Here, you can see that the actual wall thickness is wider than the reinforcement itself. We have to remedy this situation.

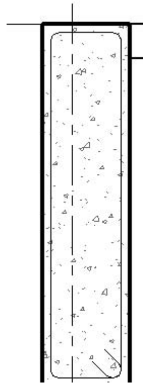
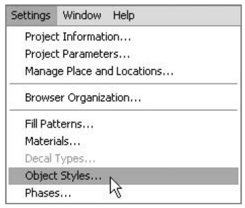


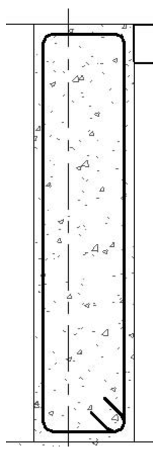
FIGURE 10.7
Choose Settings ➢ Object Styles.



These items are set to 1 by default for both projection and cut. Since a line weight of 1 is probably ridiculously thin, we recommend that you set them to at least 4 depending on your firm's standards. See Figures 10.8 and 10.9. Gaining control of object styles in Revit Structure will greatly enhance your performance.

It is also worth mentioning that if you do have to change a setting in a specific project, you'll have to make the same change for the next project. So be sure to add these settings to your template file, or contact the person in charge of your company's BIM standards. For more information on standards, see Chapter 17, which addresses all of the standards and the procedures involved. You know how you want the reinforcement to look as far as line weights are concerned.

FIGURE 10.8
The line weights are more consistent with how reinforcement is actually shown.



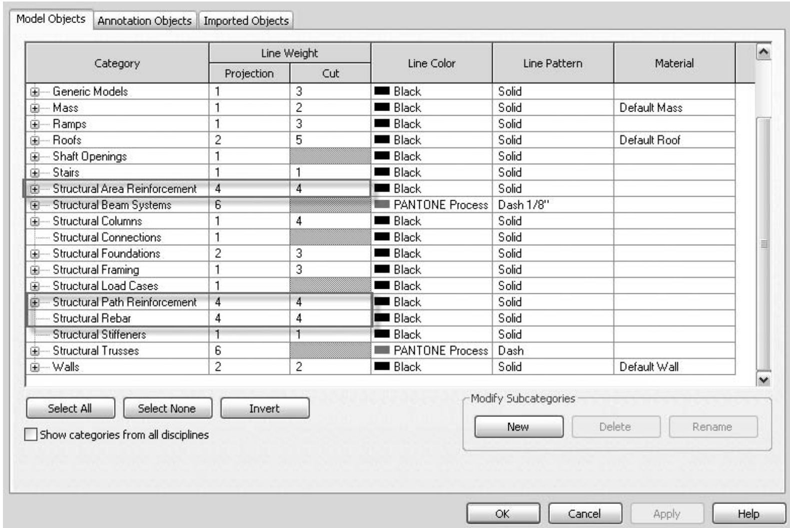


FIGURE 10.9
The Model Objects tab in the Object Styles dialog box lets you controls the line thicknesses on an object-by-object basis.

Now it is time to configure some settings that will determine the coverage that will be applied to the bars in relation to walls, slabs, and footings.

Cover Settings

As mentioned in Chapter 6, each wall type has its own cover settings. You can define the minimum setback from the face, top, or bottom of a wall or foundation that Revit Structure will allow you to automatically place reinforcement. As the rebar is graphically being placed into a wall, foundation, or slab, you will see blue alignment lines. These lines, as shown in Figure 10.10, indicate the minimum allowance of cover. Once you define the reinforcement as shown in Figure 10.11, the cover settings will serve as the guidelines for the pattern.

Cover settings are instance parameters that are associated with each wall, foundation, and slab. If you select any of these items, such as a concrete wall, and click the Element Properties button on the Options bar, you will see a Structural category. This category contains the cover settings for that specific item, as shown in Figure 10.11. The settings are contained in drop-down list to the right of the parameter. There are eight choices by default. The increments of the offsets are built into these settings.

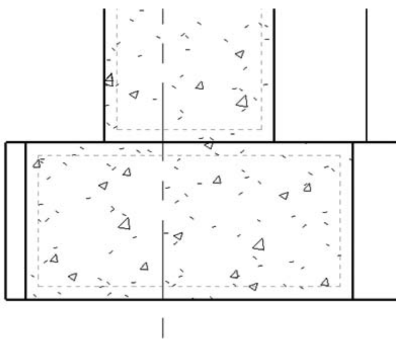


FIGURE 10.10
Notice the faint hidden lines, indicating the cover distance.

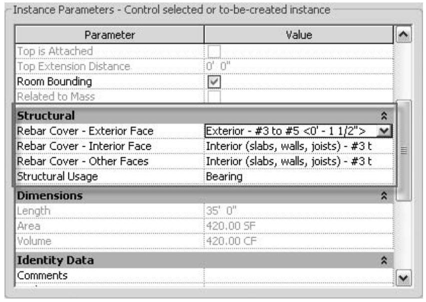


FIGURE 10.11
Instance parameters for cover settings

These eight choices are predefined settings built into the Revit Structure default template. You can add or subtract from these items as you see fit, as shown in Figure 10.12. Choose Settings ➢ Rebar Cover Settings to access these options.

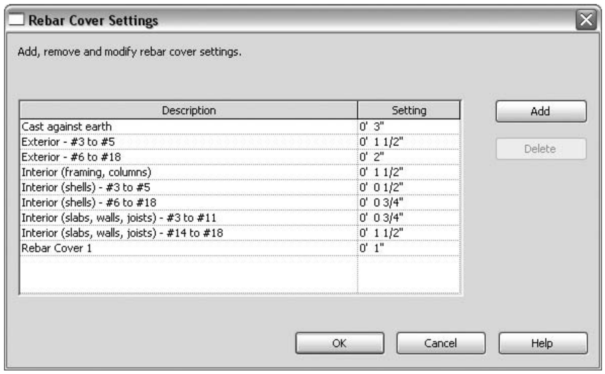


FIGURE 10.12
You can access the Rebar Cover Settings dialog box by selecting Settings ➢ Rebar Cover Settings.

Simply click the Add button, and you are off to adding a new setting. This is one instance where the out-of-the box Revit Structure settings are sufficient…almost. Also, we recommend that as you are creating these new settings, keep in mind the fact that you should not overdo it. Yes, some projects will require more cover situations than others, but give careful consideration to what settings should go into the templates, and which settings should be specific to this project. Once these settings are set, you are ready for the next step: adding the bars to the model.

Adding Bars Parallel to the Current Work Plane

As sections are cut, and the detail views start to populate the Project Browser, you may notice that when you select a wall, a footing, or a slab, additional icons appear in the Options bar as shown in Figure 10.13.

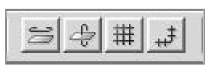


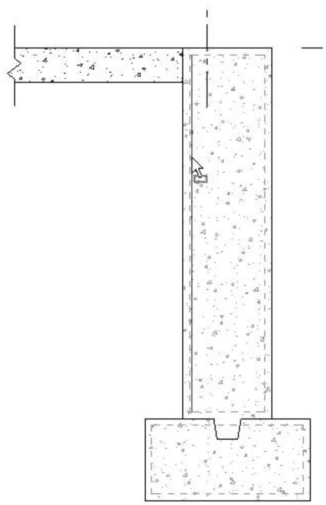
FIGURE 10.13
The placement options that appear on the Options bar

Each of these items allows you to place reinforcement in an item, if you are in the proper view. For example, if you are looking at a section of a foundation wall, you will select the wall that you wish to add reinforcement to. The placement options appear. If you are placing a rebar lengthwise, you select the Place Rebar Parallel to the Current Work Plane, as shown in Figure 10.14.

FIGURE 10.14
The Place Rebar Parallel to the Current Work Plane option found on the Options bar

As you hover your cursor over the wall, you will see dashed blue alignment lines, as shown in Figure 10.15. This is Revit Structure telling you the farthest you can get to the face of an item before the rebar cover settings disallow the placement. If you try to exceed this plane, Revit Structure will disallow the placement. If you click right on the blue alignment line, Revit Structure will place the rebar into the model. The top and the bottom of the rebar will start and terminate at the specified cover settings.

FIGURE 10.15
Placing a bar parallel to the current work plane



With the rebar in place, you can now select it and view the Options bar. Notice the placement option that allows you to specify a layout, as shown in Figure 10.16. This single bar can be set to run the length of the wall at the spacing specified in the options.

After the spacing has been set, you may notice that Revit Structure will keep a running count (quantity), as shown in Figure 10.16. Get used to having this kind of information right in your front pocket!

Adding reinforcement in Revit Structure is great. Once the rebar is placed in the model, you can then configure its visibility state based on view.

Figure 10.16
Setting the layout to Maximum Spacing at an increment of 12″ O.C.

Rebar Visibility States

One issue with reinforcement in Revit Structure has always been that rebars will show up in views where you didn't want them to. Each piece of reinforcement has its own instance parameter, which will control the appearance of reinforcement as it relates to other views. To gain access to the settings for the visibility states of a specific reinforcement item, you must select the rebar to be examined. When the Options bar appears, click the Element Properties button. In the Element Properties dialog box, you will see a Graphics category. In the Graphics category, you will see a View Visibility States parameter, as shown in Figure 10.17. Once you click the Edit button, you will then be able to control exactly where this specific reinforcement will appear.

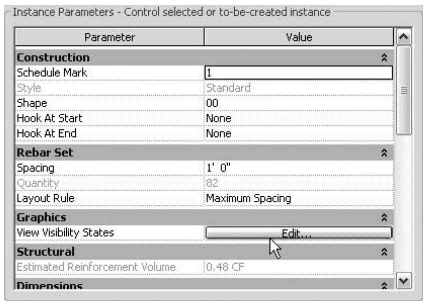


Figure 10.17
Select a rebar, and click the Element Properties button. This will allow you to view the visibility states for the rebar.

In the Rebar Element View Visibility States dialog box, you will see most of the views are unchecked, as shown in Figure 10.18. The checks do not actually turn the bars off—they merely allow the bars to be obstructed by the body of the host element. When viewed unobstructed, these bars will show through the host walls, foundations, and slabs. By default, the default views that show reinforcement unobstructed are sections. Typically, plans and elevations have rebars obstructed by the host.

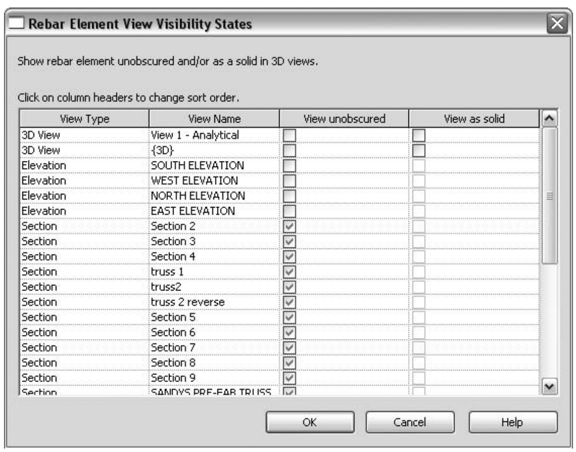


Figure 10.18
Rebar Element View Visibility States dialog box

Similar to adding bars parallel to the current work plane, you can add horizontal bars that run perpendicular to the current work plane in the same view.

Adding Bars Perpendicular to the Work Plane

Using the same procedure as adding parallel bars, you can place bars that are perpendicular to the work plane. In a sectional view, as illustrated earlier in Figure 10.15, these bars are considered horizontal bars. To add these bars, simply select the wall again. You can then click the Place Rebar Perpendicular to the Current Work Plane button. By default, these bars will come in based on a horizontal spacing. As you are placing the bar, press the spacebar to flip the orientation to a vertical plane. As you place the bar, the same cover planes will appear, constraining the bar's placement. If you place the horizontal bar at the top of the wall, it will not exceed the cover set for the wall's top, as shown in Figure 10.19.

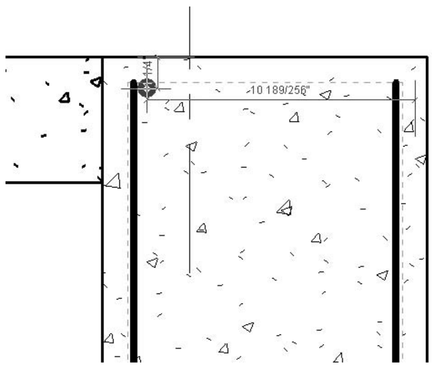


FIGURE 10.19
Adding bars perpendicular to the current work plane

Once the bar is placed, you can then select it and alter the layout and the spacing. If you change the layout to Maximum Spacing and change the spacing to 12″, the bars will array down the wall to the bottom cover, as shown in Figure 10.20. You can then simply mirror the bars to the opposite face of the wall once they are in place.

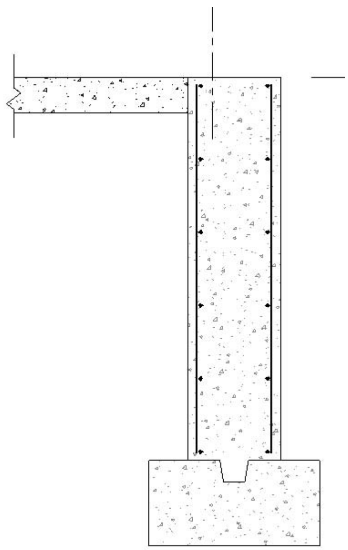


FIGURE 10.20
Horizontal bars at 12″ O.C. maximum spacing

The same procedure can be applied to footings and slabs as well. Once you have mastered applying reinforcement to one item, you have mastered them all!

The Rebar Shape Browser

As mentioned earlier, many of the advancements in the Revit Structure 2009 application have occurred in the reinforcement area. The biggest improvement is what is called the Shape Browser. As you select an item to place reinforcement in, you then select if you want the reinforcement perpendicular to the current work plane, or parallel to the current work plane, as illustrated earlier in Figure 10.13. Once you determine the direction of the bars, you will see a list on the Options bar. It allows you to select a rebar shape and lists a type, as shown in Figure 10.21. To the right of the list is a builder button (which shows an ellipsis). This button will turn the rebar Shape Browser on or off.

FIGURE 10.21
The Shape list on the Options bar

With the rebar shape browser turned on as shown in Figure 10.22, you will see that many default choices are built into the program.

> **HEY! WHERE ARE MY BARS?**
>
> If you are in a model that was created before Revit Structure 2009, you will see only one shape available. If that is the case, you can choose File ➢ Load from Library ➢ Load Family. In the default Imperial library you will see a Rebar Shapes folder. Click in the Rebar Shapes folder and then press Ctrl+A. This will select all of the rebar shapes. Click the Open button, and all of the default Revit rebar shapes will be loaded into your model.

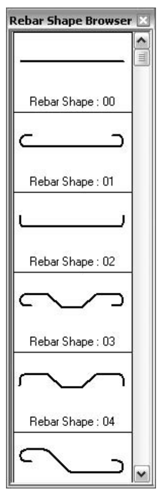


FIGURE 10.22
The Shape Browser allows you to add any rebar shape to your model.

As you find a shape and place it into the model, you will see that the rebar cover settings are, again, dictating the height and width of the bar configuration, as shown in Figure 10.23.

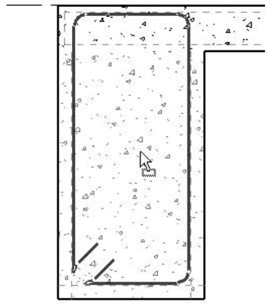


Figure 10.23
Placing a rebar shape from the browser

Rotate That Stirrup

As you are placing shapes with ties or other lap geometry, you can press the spacebar to rotate where the lap or tie will be positioned before you place the bar configuration.

Once the bars are placed, you can then select them. Notice there are quite a few grips and shape arrows. These provide you with the ability to freely change the bars to a new location if the default position is not acceptable. See Figure 10.24.

Once the bar is in place, you can change the spacing and the view settings. If you want the bars to display in 3D as a 3D solid, open the element properties and change the visibility settings to View Unobscured in {3D}. You can also check View As Solid. Refer again to Figure 10.18.

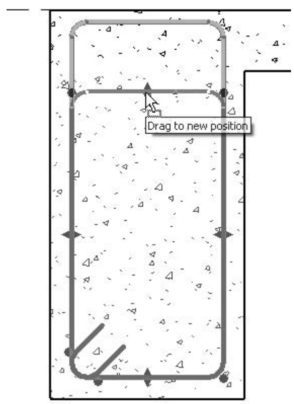
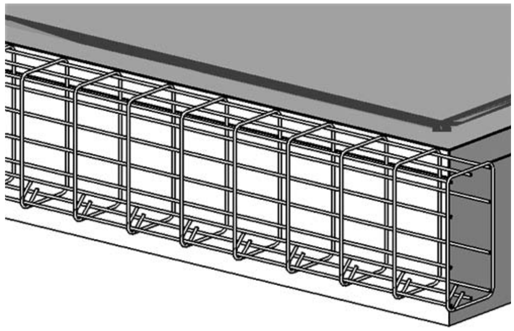


Figure 10.24
Positioning the reinforcement

You place bars that are perpendicular to the current work plane in the same manner. Select the item you want reinforced, and click the Place Rebar Perpendicular to the Current Work Plane button on the Options bar. In the Shape Browser, make sure you have the bar you want, and simply place the bars in the model just as you would if you were placing blocks. Remember to set the visibility state to show unobstructed in the 3D view if so desired. See Figure 10.25.

Figure 10.25
Showing the reinforcement in 3D as unobstructed



Revit Structure, as you can see, allows a tremendous amount of flexibility in terms of allowing you to model rebar as efficiently as possible, while still maintaining a true 3D modeling environment. This procedure works quite well until you come to a situation where the bars provided by Revit Structure do not provide enough choices to model a specific situation you may have. It is here that Revit Structure allows you to sketch your own rebar shape and add it to the project.

Sketching Rebar

The first time you start using the new Shape Browser, you will discover that there is no way Revit Structure could possibly have every shape required—especially in situations where one component needs to be doweled into another. To sketch rebars in a wall, slab, or floor, simply select the component that is to receive the reinforcement. On the Options bar, select Place Rebar Parallel to Current View. Then on the Options bar, click the Sketch button. Revit Structure will prompt you to select the item you want to place the reinforcement in again. Once you select the object, you can then freely sketch the reinforcement. This is effective for doweling a foundation wall into a footing, as shown in Figure 10.26. You may notice that as you are drafting, if you have a 90-degree corner, Revit Structure will add the bend radius for you. Revit Structure knows you are drawing a rebar, so it stands to reason that things like this will be automatic.

Real World Scenario

Look Out for the Hole!

The ability to sketch rebars around a slab or wall opening is indispensible. Many times, we have faced reinforcement of massive openings created in demolition projects where the rebar needed to be carefully placed and counted around these openings. The ability to draw the bars, and space them so there was a bar counted on both sides of the wall, dramatically increased the accuracy of the quantity takeoffs and even reduced the time required to do the drafting.

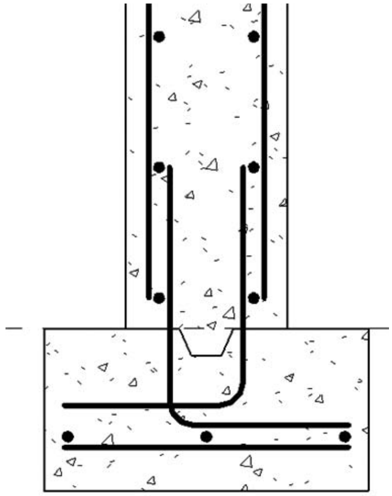


FIGURE 10.26
Because these footing dowels are a nontypical Revit Structure dowel, you can see that the free sketch method was essential in this situation.

One nice feature appended to the new rebar modeling tool is that it will automatically add the new rebar sketch to the Shape Browser for future use in this model, as shown in Figure 10.27.

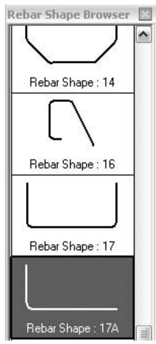
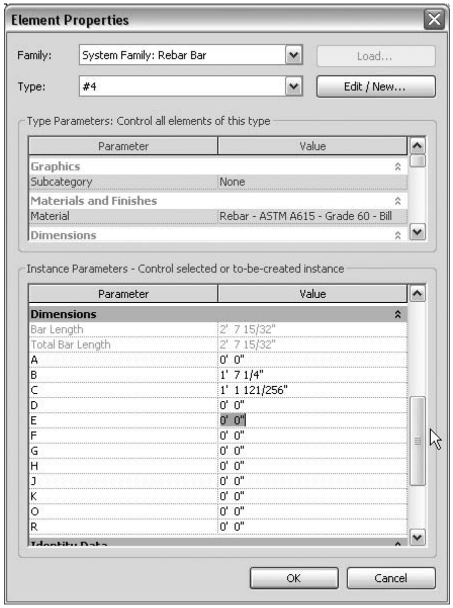


FIGURE 10.27
The newly added rebar shape

The new rebars that you create have the same intelligence as the Revit Structure–provided reinforcement. If you select the rebar you create, you will see in the Options bar that you can still specify a unique spacing for each instance of the reinforcement. Also, if you select the new reinforcement and click the Element Properties button, you will see that you can change the # type, as well as the dimensions (A through R), as Figure 10.28 shows.

Although it seems we have explored all the methods there are for placing reinforcement into a model, there are more to go. Suppose you wanted to reinforce an entire slab in one shot? Or perhaps there are perimeter conditions you need to add to the model. Revit Structure will allow you to do these tasks with commands specific to each situation.

FIGURE 10.28
The rebar's Element Properties dialog box allows you to edit any dimension, such as the dimensions labeled A through R.

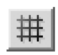


Area Reinforcement

With the Area Reinforcement command, Revit Structure will allow you to place rebars in an entire slab or wall in one action, thus eliminating the need for toiling over exact placement and configuration.

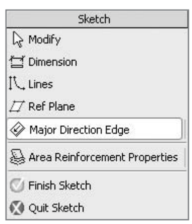
To place area reinforcement in a slab, you will need to be in the plan that the slab occupies. Once there, you select the slab that is to be reinforced. Once you do, the reinforcement icons appear on the Options bar. Click the Sketch Area Reinforcement button shown in Figure 10.29.

FIGURE 10.29
The Sketch Area Reinforcement button on the Options bar

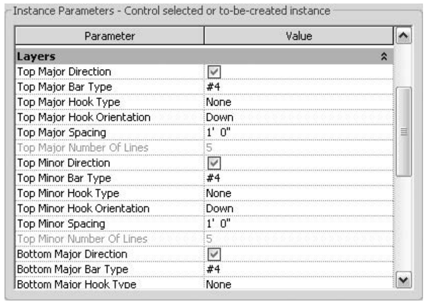


This command will put you in Sketch mode. On the Options bar, you can use the Pick Lines button and start clicking the edges of the slab. The first line you click will have two shorter lines on either side. This indicates the slab direction. If this is incorrect, you can change the direction by clicking the Major Direction Edge button on the Sketch bar, as shown in Figure 10.30.

FIGURE 10.30
The Sketch bar to aid in the placement of area reinforcement.



Another essential button in the Sketch panel is the Area Reinforcement Properties button. This opens a dialog box that will gain you access to the configuration and type of reinforcement that is going into your slab. A nice thing about the properties in area reinforcement is that you can specify that the reinforcement have two different layers (with separate properties for the type and spacing of the major and minor bars in each layer), as shown in Figure 10.31. Each of these layers can be turned off if one or the other is not needed (see Figure 10.32).



Instance Parameters - Control selected or to-be-created instance

Parameter	Value
Layers	
Top Major Direction	☑
Top Major Bar Type	#4
Top Major Hook Type	None
Top Major Hook Orientation	Down
Top Major Spacing	1' 0"
Top Major Number Of Lines	5
Top Minor Direction	☑
Top Minor Bar Type	#4
Top Minor Hook Type	None
Top Minor Hook Orientation	Down
Top Minor Spacing	1' 0"
Top Minor Number Of Lines	5
Bottom Major Direction	☑
Bottom Major Bar Type	#4
Bottom Major Hook Type	None

FIGURE 10.31 Specifying the layer configuration in the element properties of an area reinforcement

In the Element Properties dialog box, you are also given the choice to change the additional top and bottom offsets. These will be added to the cover settings, and Rivet Structure will place the reinforcement at the specified offset. For example, if the cover setting for the slab is 1½″ down from the top face, and you specify an additional offset of 1″, the rebar will be placed 2½″ below the top face. This can be handy if there is a finish or a topping you wish to allow for.

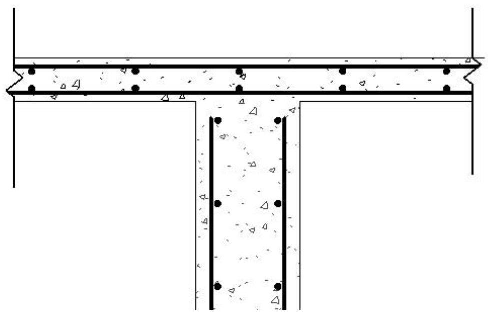


FIGURE 10.32 Area reinforcement in the slab section shows both top and bottom bars (remember this was added in a plan view).

In plan, the area reinforcement will automatically be tagged. This behavior can be changed, and a different custom tag can be specified. Select Settings ➢ Annotations ➢ Loaded Tags. In the resulting dialog box, you'll see a category for Structural Area Reinforcement, as shown in Figure 10.33. This is the tag that Revit Structure will automatically apply to the area reinforcement in plan.

The tag will appear automatically, as shown in Figure 10.34. If you do not want to tag the item, you can delete the tag—it will not affect the actual reinforcement.

Area reinforcement is perfect for reinforcement structural slabs, but it can be translated to walls as well. The procedure is almost exactly the same.

FIGURE 10.33
The Structural Area Reinforcement category with the automatic Area Reinforcement Tag

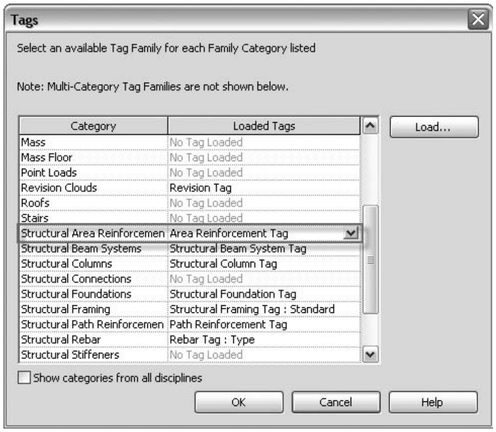
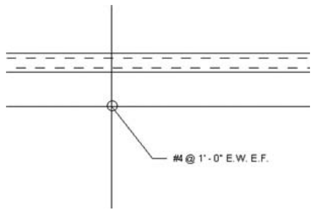


FIGURE 10.34
The automatic plan tag that Revit Structure provides as you place area reinforcement



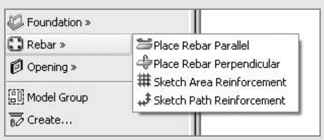
AREA REINFORCEMENT FOR WALLS

Placing area reinforcement in a wall is sometimes preferred over having to place bars individually and then having to specify the spacing. Each method can have its advantages, and after you know how to do both, you can determine which process best suits you.

To place area reinforcement in a wall, you must first open an elevation that is looking straight at the wall that is to be reinforced.

THERE IS A DIFFERENT WAY

This chapter has enforced the procedure of selecting the item that will receive the reinforcement, then clicking the proper button on the Options bar. You can, however, go to the Modelling tab and select the reinforcement choice from there as well, as shown in the following illustration.

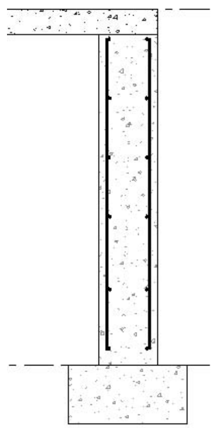


In the elevation, you can select the wall, and click Area Reinforcement on the Options bar. This will put you back in Sketch mode. At this point, you can sketch the perimeter of the reinforcement, similar to placing the reinforcement in the slab. Make sure you do not exceed the actual boundary of the wall when doing this (see Figure 10.35).

FIGURE 10.35
The area Reinforcementsketch profile. The double lines on the left sketch line indicate the major direction of the bars.

Once the sketch is finished, you can click Finish Sketch on the Design bar. If you cut a section through this wall, you will see that the reinforcement is laid out as expected, as shown in Figure 10.36. If the bar layering is reversed, you can go back to the elevation, select the area reinforcement, and click Edit. In the resulting dialog box, you can change the Major Direction setting.

FIGURE 10.36
Area wall Reinforcement as shown in section



Now that the mass areas are reinforced, we can move on to reinforcement of specific areas, such as providing additional dowels to the perimeter for the slab by adding what Revit Structure refers to as *path reinforcemen*.

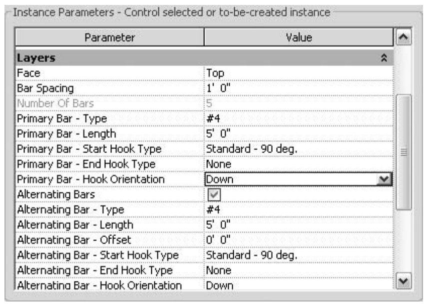
Path Reinforcement

There are occasions where you are going to need to reinforce the edges of a slab independent of area reinforcement to prevent the slab edges from curling. This is where the Path Reinforcement tool comes in handy. Like the other Revit Structure Reinforcement tools, you apply path reinforcement to a slab by first selecting the slab that needs the reinforcement, and then clicking the Path Reinforcement button on the Options bar, as shown in Figure 10.37.

FIGURE 10.37
The Path Reinforcement button appears on the Options bar after you select the slab you wish to reinforce.

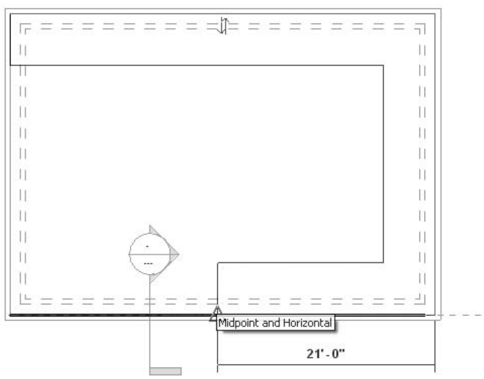
While in the Sketch mode, it is a good idea to go to the Element Properties dialog box for the path reinforcement to configure the bars as shown in Figure 10.38. Within the dialog box, you can specify in which direction the bars will hook. By default, the bars are hooking up. Normally in a slab bearing situation, you would need to change the hook direction to down so that the bars can hook into the top of the bearing wall. Also, you can turn on alternating bars. This allows you to stagger the bars at whatever increment you have specified. The hooks are specified at each end of the bar.

FIGURE 10.38
Instance parameters for edge reinforcement allow you to further control the function of the bars, such as primary spacing and the addition of alternating bars.



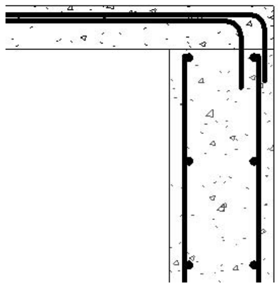
Once you are finished specifying the properties, you can click OK to return to the view. From there, you can click the Lines button on the Sketch panel, and draw the perimeter of the reinforcement, as shown in Figure 10.39.

FIGURE 10.39
Sketching the path reinforcement



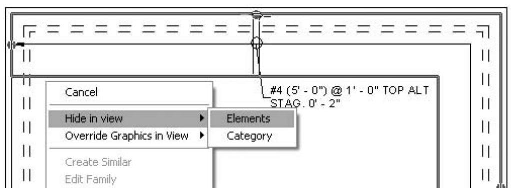
Once you've sketched the path, you can click Finish Sketch and Rivet Structure places the reinforcement into the slab. It is a good idea to cut a section through the path reinforcement and look at the spacing and the bar configuration, as shown in Figure 10.40. If it is not as expected, you can select the bars in that view, and click the Element Properties button in the Options bar. You can then configure the properties without having to go back into Sketch mode. Figure 10.40 shows the reinforcement with a 2″ offset for the alternating bars.

FIGURE 10.40
The bars in section



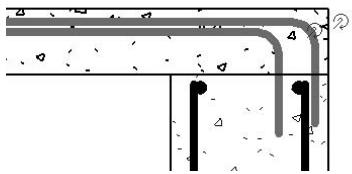
The plan view will show the depth of the bars as well as the tags and symbols. You may not want to show all of this. You can select the path reinforcement line and right-click. From the context menu select Hide in View ➤ Elements, as shown in Figure 10.41.

FIGURE 10.41
Hiding the path reinforcement in view



Another item to mention in all reinforcement is the end hook configurations, as shown in Figure 10.42. You can override the end hooks of any reinforcement by simply selecting the bars and clicking the end hook icon that becomes visible. Many times this is an easier method than trying to specify an end hook condition in the properties dialog box. You may find it necessary to be able to graphically see in which direction the bars are going. It is also sometimes hard to determine which end of the bar you are trying to configure. This on-screen toggle makes the process much easier.

FIGURE 10.42
End hook configuration shown with the dowels extending into the top of the foundation wall



Compared to the simple drafting performed in AutoCAD, adding reinforcement to a Revit Structure model is not harder but is certainly different. With the various options offered by Revit to aid in the placement, this process is obviously beneficial. Also, there is a downstream benefit. As you will see in Chapter X, reinforcement can easily be scheduled and quantified after it has been placed in the model. This is a tremendous benefit. Keep in mind, however, that if you are to provide scheduling and quantities you must be diligent in the bars' placement. You can only expect Revit to give you back what you put into the model.

Exercise: Using Revit Reinforcement

In this exercise, you can either use the Revit Structure model provided at the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`, or you can follow along with your own model. The object of this exercise is to get you comfortable using the Revit Structure reinforcement, and to make you never want to go back to drafting again. To start, follow these instructions:

1. Open the file called `reinforcement model.rvt` from the book's companion website. You may substitute your own model if you want.
2. The first procedure is to configure the settings so the reinforcement looks correct. To do this, choose Settings ➢ Object Styles.
3. Starting from the top of the list, change the following line weights:

    Floors: 2,2 (the first is projection, the second is cut)

    Structural Area Reinforcement: 4,4

    Structural foundations: 2,2

    Structural Framing: 1,2

    Structural Path Reinforcement: 4,4

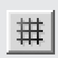
    Structural Rebar: 4,4

    Walls: 2,2

4. Click OK to close.

The next order of business is to place area reinforcement in the slab. To do this, use the 1ST FLOOR level view and select the slab. You may have to select an area around the corner of the model and click the Filter button on the Options bar. Once in the filter, select only Floors and click OK.

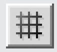
1. In the Options bar, select the Sketch Area Reinforcement button, shown here:



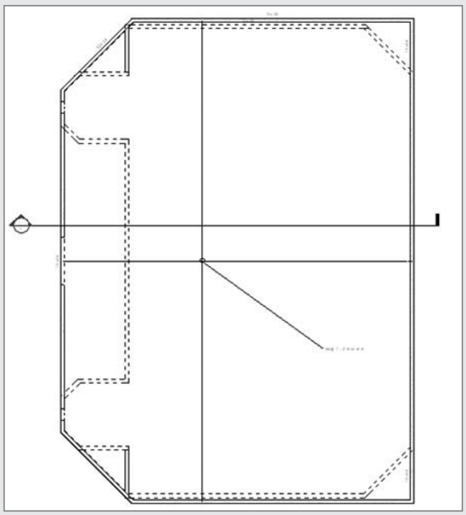
2. In the Sketch panel, click the Lines button. (It should be on by default.)
3. On the Options bar, click the Pick Lines tool.
4. Select all the outside edges of the slab. Remember that you must have a completely closed loop with no gaps or overlapping lines.

The next order of business is to place area reinforcement in the slab. To do this, use the 1ST FLOOR level view and select the slab. You may have to select an area around the corner of the model and click the Filter button on the Options bar. Once in the filter, select only Floors and click OK.

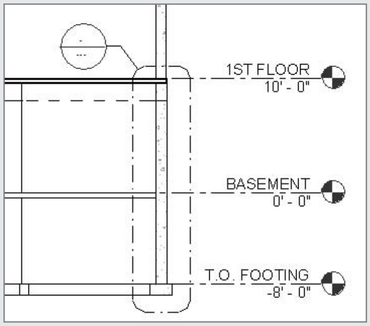
1. In the Options bar, select the Sketch Area Reinforcement button, shown here:



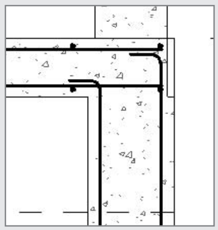
2. In the Sketch panel, click the Lines button. (It should be on by default.)
3. On the Options bar, click the Pick Lines tool.
4. Select all the outside edges of the slab. Remember that you must have a completely closed loop with no gaps or overlapping lines.
5. Click the Area Reinforcement Properties button.
6. Verify the Layout Rule is set to Maximum Spacing.
7. Keep the rest of the defaults the same. Remember, you can always go back and change the properties after you review a section through the model.
8. Click OK. Then click Finish Sketch to complete the placement.
9. Revit Structure will place the rebar into the slab. Create a section through the slab, as shown here:



10. Create a callout of the right side of the section where the 12″ wall/beam meets the 10″ wall extending down to the footing, as shown in the following graphic. Make the callout's scale 1/2″ = 1′-0″.



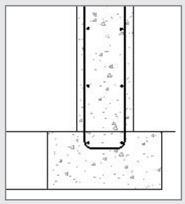
11. Select the 12″ concrete bearing wall.
12. Click the Place Rebar Parallel to the Current Work Plane button on the Options bar.
13. In the Shape Browser, select Rebar Shape: S5. Make sure the hooks are to the top and pointing to the left. Select the rebars, and grip-stretch them into the slab, as shown in the following illustration:



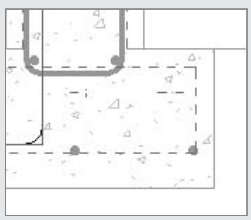
Now that the bars are "roughed in," you can alter their behavior by accessing the element properties.

1. Select the new rebar, and click the Element Properties button.
2. For the Rebar Set category, set the Layout Rule to Maximum Spacing, and set the Spacing to 1′-0″.
3. Click Edit/New.
4. Click Duplicate.
5. Name the rebar **`#4 with 12″ hooks`**.
6. Under Hook Lengths, click the Edit button.
7. Make the Stirrup/Tie –90 degree 1′-0″.
8. Click OK three times.

9. Select the same foundation wall.
10. On the Options bar, click the Place Rebar Perpendicular to the Current Work Plane button.
11. Select Rebar Shape: 00.
12. Press the spacebar once to orient the bar vertically.
13. Place the bar at the top cover alignment line and to the right of the vertical bar.
14. Make sure the Layout setting is Maximum Spacing and that the Layout Rule setting is 12″ = 1′-0″.
15. Mirror the bars to the other side.
16. Pan down to the where the wall bears on the footing.
17. Select the S5 bar and grip-stretch it out of the footing.
18. Select the horizontal bars and do the same, as shown in the following illustration:



19. Select the footing, and click the Place Rebar Perpendicular to the Current Work Plane button.
20. Select Rebar Shape: 00.
21. Place the rebar to the bottom left of the footing in the corner of the cover alignments.
22. Change the Layout setting to Fixed Number.
23. Change the Quantity setting to 3.
24. Select the 12″ foundation wall.
25. Click Place Rebar Parallel to the Current Work Plane.
26. On the Options bar, click the Sketch button.
27. Select the wall again.
28. Draw in a hooked dowel extending from the second bar into the footing, as shown here:

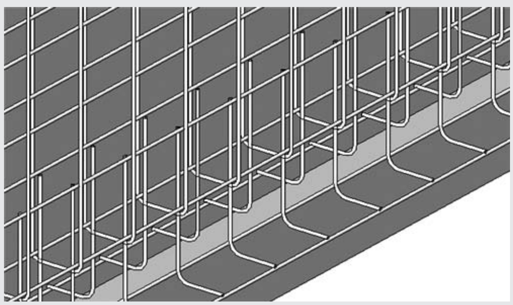


29. Click Finish Sketch. Notice that the dowel is now added to the Shape Browser.

You can now perform common drafting tasks to repeat the efforts without going through the element properties repeatedly.

1. Mirror the dowel.
2. Change the Spacing setting to Maximum Spacing.
3. Change the spacing to 12″ = 1′-0″.
4. Select all of the rebars you have put into the model.
5. On the Options bar, click Element Properties and then click the View Visibility States button.
6. Select View Unobscured for All {3D} Views.
7. Select View as Solid.
8. Click OK.
9. Display the default 3D view.
10. Change the view Detail Level setting to Fine.
11. Change the Model Graphics Style setting to Shading with Edges.
12. Click the Show Thin Lines button.

You can now see the massive amount of steel that goes into a wall like this. All are generated using a section view. It is good to keep in mind that this section was generated by using tools that are no more difficult than those you'd use in actual drafting. As a matter of fact, most of these tools are much easier than those you'd use in standard drafting. You do not have to add a radius to bends, and you can clearly specify hook lengths as well. As you will see in the next chapter, this reinforcement can be put into a quantity take-off and a reinforcement schedule. As you learn to draft, you will see that this reinforcement can be tagged automatically, allowing you to maintain consistency throughout the project, as shown here:



The Bottom Line

Draft a 2D rebar. In Revit Structure you can simply draft reinforcement by configuring line styles to be similar to an AutoCAD environment.

Master It Creating line work in Revit Structure is similar to drafting in AutoCAD. Name the process involved in getting the correct line weights and adding the lines to the model.

Configure rebar settings. In Revit Structure you can place reinforcement as actual objects as opposed to simple drafting. To do this correctly, however, you need to extensively configure the rebar settings for both graphics as well as performance.

Master It Walls, footings, and slabs have cover settings that allow you to place reinforcement in a more organized and accurate approach. How is this done?

Draft a 3D rebar. Although Revit Structure uses a modeling approach, it is often necessary to be able to sketch reinforcement first, and then add it to the 3D Shape Browser once it is completed.

Master It Placing 3D reinforcement can be done in two different ways. Describe both.

Add rebar shapes. By default in Revit Structure 2009, you have a multitude of reinforcement to choose from. These shapes are preloaded into the template file you are using. Revit Structure allows for the importing of additional shapes.

Master It You may be working in a model that was created before the Revit Structure 2009 version release. The model will not have any rebar shapes. How do you import the shapes?

Chapter 11

Schedules and Quantities

Now that you are more-or-less comfortable with modeling in a 3D environment, it is time to delve into the often talked about, but never really explained, 4th dimension. That's right. The 4th dimension. This is the non-graphical side of Revit Structure that allows us to keep track of all of these components that comprise our model.

Often times you are going to ask yourself, and even be asked "Why are we using BIM over standard drafting?" We were making money before…why take the risk? This author has asked this question to himself, and of course has been asked as well. The answer, as it pertains to the topic of this chapter, is simple.

Every effort made in modeling your building starts to be rewarded by mining the vast amount of data you have created. The fact that you can take 5 minutes to create a comprehensive steel quantity takeoff that is accurate and will flex along with the model is priceless. To take this 5 minute process one step further, typically a firm is going to have a pre-defined grouping of schedules and quantity takeoffs already defined and ready to go before the first beam or column is placed in the model. This means that during the life of the project, these reports can be printed or viewed literally by the push of a button. Not bad for having to do nothing more than simply place items into a model.

In this chapter you will learn to:

- Create Schedules
- Create Material Takeoffs
- Create Legend Schedules
- Send Schedules to Microsoft Excel

Scheduling Basics

To begin creating a schedule, you do not have to have a "populated" model to start. Many times schedules are created in the company template. This is done with the intent that the designer only has to engage in the modeling process while the schedules and quantity takeoffs are being "built" in the background. This is great however, the term "design" means "to create." Every job has some kind of defining item that is different, and will certainly have different requirements based on the output data. Or, in many cases, schedules are used to simply keep track of your design. This is possible because of the fact that a schedule is a two way street. Any changes in the model will immediately update the schedule. Any changes in the schedule will immediately update the model. For example: if there is a group of beam sizes you wish to change in a certain area, you can open the schedule, locate the beam sizes and change them right in the schedule. This updates not only the schedule you are currently in, but the plans, elevations and sections.

This is the typical reaction to Revit. If you make a change in one place, other areas of the model are going to be influenced by these changes.

To create a schedule, you can go to one of two different places to start. The first place is in the View tab on the Design Bar as illustrated in Figure 11.1.

FIGURE 11.1 Locate the Schedules/Quantities button on the View tab.

Another place you can go to find the Schedule/Quantities dialog is View ➢ New ➢ Schedule Quantities as illustrated in Figure 11.2.

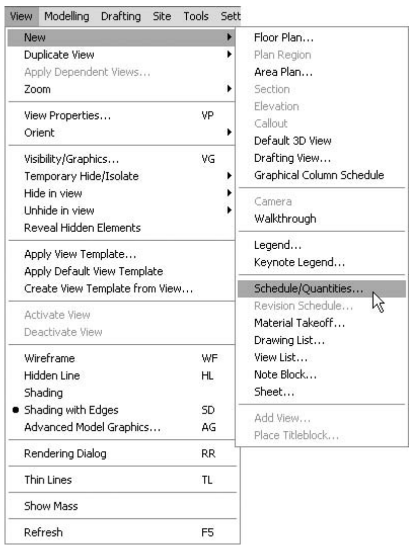


FIGURE 11.2 Finding the Schedule/Quantities dialog in the File Menu

Once you are in the Schedules/Quantities dialog it suddenly becomes apparent that there is a multitude of choices. At this point, you make the decision: is this schedule going to on my drawing sheets, in an estimate, or just for my own use as I model my building? Whatever the choice is, there are five basic categories involved in creating a schedule.

Fields: This determines what categories will be added to the schedule

Filter: This determines which fields will actually show in the schedule

Sorting/Grouping: This determines how the chosen fields will be arranged and grouped

Formatting: This determines how the data will behave in terms of mathematical computations, and how the data is formatted

Appearance: This determines how the schedule will look in terms of font types and sizes. The border is addressed here as well.

Item counts on the fly!

If the choice is to use a Multi-Category schedule, you are choosing to schedule every building component in the model. Having a Multi-Category schedule in the background, *numerically* keeping track of your model, can become a fantastic reference as you design your structure. It may never see the "light of day" as far as putting the schedule on a sheet is concerned, but it is always there to refer to.

Schedule Properties

After you start the Schedules/Quantities command, you are given choices in the left field as illustrated in Figure 11.3. If, for example, you would like a schedule listing all of the structural framing in the project, you can select Structural Framing from the Category list on the left side.

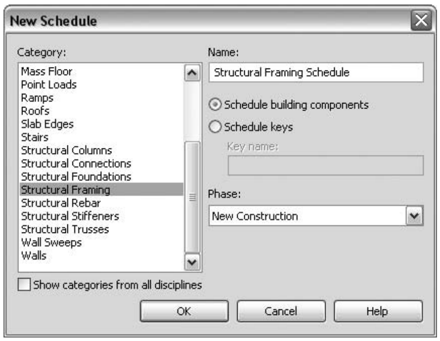


Figure 11.3
Choosing a category

Once the category is chosen, you can click the OK button. This will bring you to the Schedule Properties dialog as illustrated in Figure 11.4. Once this dialog opens, you will see a series of tabs across the top of the dialog. The first tab you will see is the Fields tab. It is the default tab as you start a new schedule for a good reason. You obviously need to have some fields in which to tell Revit Structure what to base the schedule on.

Fields

The fields tab is broken down into two columns. The column to the left (Available Fields) is a list of all of the available fields in the chosen category. For this example the chosen category is Structural Framing, so therefore the choices are pursuant to this item. The column to the right

(Schedule Fields) is a list of the fields that are taken from the left. To add a field, you simply choose an available field from the left column, and click the Add button as shown in the graphic below.

The field is now in the column to the right, in the order in which it will be displayed in the completed schedule. See Figure 11.4.

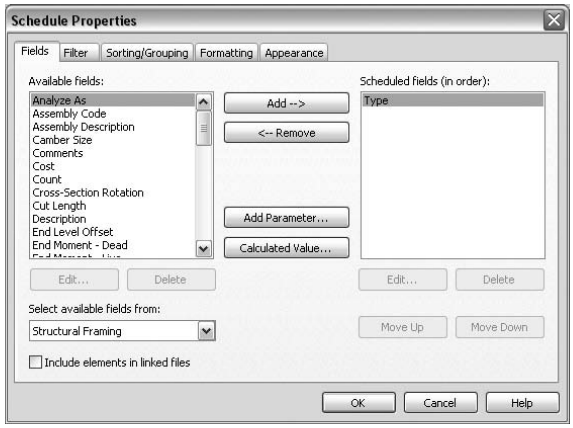


FIGURE 11.4
Adding a field

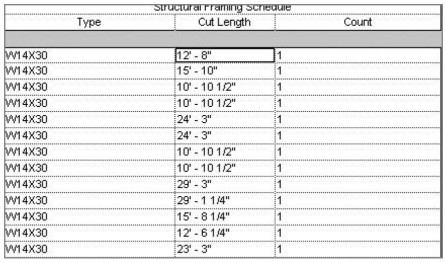
Once a field is added to the Schedule Fields column, the Remove button becomes available. (You will probably use this button almost as much as you use the Add button).

While adding items to the Schedule Fields column, you do not have to worry too much about the order in which you are adding them. Below the Schedule fields column you will see buttons which allow you to move the fields up or down, as illustrated in Figure 11.5.

FIGURE 11.5
You can move any field up and down in the Schedule Fields column

Once the desired fields are moved to the Schedule Fields column, you can click OK. This will give you a preview of the new schedule, as illustrated in Figure 11.6.

FIGURE 11.6
The schedule view

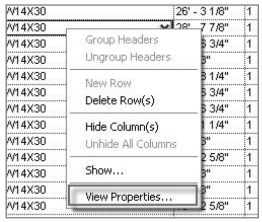


Structural Framing Schedule

Type	Cut Length	Count
W14X30	12' - 8"	1
W14X30	15' - 10"	1
W14X30	10' - 10 1/2"	1
W14X30	10' - 10 1/2"	1
W14X30	24' - 3"	1
W14X30	24' - 3"	1
W14X30	10' - 10 1/2"	1
W14X30	10' - 10 1/2"	1
W14X30	29' - 3"	1
W14X30	29' - 1 1/4"	1
W14X30	15' - 8 1/4"	1
W14X30	12' - 6 1/4"	1
W14X30	23' - 3"	1

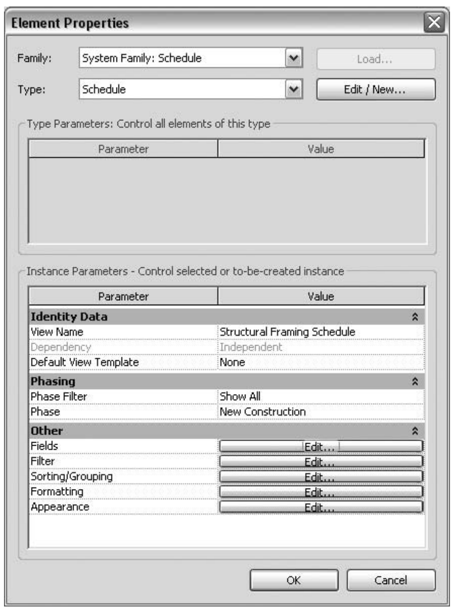
To get back to the Schedule Properties dialog, you will need to right-click in any field. From there you can select View Properties, as illustrated in Figure 11.7.

FIGURE 11.7
Right-click to go back to View Properties



After you click the View Properties choice, you will be brought to the Element Properties dialog which will list the available tabs for the Schedule Properties, as illustrated in Figure 11.8. Click on the long Edit button to the right of the Fields row, and this will put you back in the Schedule Properties dialog.

FIGURE 11.8
Element Properties



Just because this is simply a list of items, it does not mean that it is not live. If the field is a editable cell, you can click into it and change the information. This interaction is a two way exchange. If you change a value here, you are changing it in the model. Simply telling you to be careful here is the understatement of the century. Make sure any changes made in a schedule are deliberate and accurate.

Because Revit Structure is a database, the act of creating a schedule is identical to creating a query. You tell Revit what you want to see and how you want to see it. Revit then displays that information. For example, if you wanted to find all of your structural framing based on Type, Cut Length and Count, you would add these three items to the column on the right as shown in Figure 11.9. Once you are set on a few fields you can then proceed to run a filter. A filter, as mentioned before, is nothing more than a query in a database similar to what you would do in Microsoft Access or any database application.

Filters

The second tab in the Schedule Properties dialog is Filter. Of course, the more fields you have available, the more powerful this feature becomes. This is a great way to "weed out" items that could be problematic.

Real World Scenario

Filtering Beams Based on Length

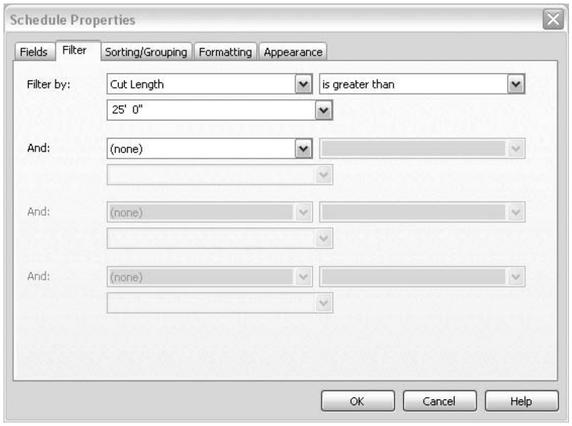
In some cases, a project could lack dimensional information. A schedule filter is then used to find beams spanning more than a specific length. Once these beams appear in the schedule, you can then move them up to a larger size. This has helped to keep track of possible problem areas that are reportable and printable for anyone to review, including project managers.

To access the filters, you can simply click on the Filter tab as illustrated in Figure 11.9. In the Filter By: category, you can then select the field in which you would like to run the filter on. This is essentially telling Revit that you only want to see the items containing this information. Any other framing items in the model that don't contain this information will be excluded from this schedule.

To the right of the Filter By: category is a field which allows you to add a condition. For the example of finding all of the framing members that exceed 25′-0″, you can select Greater Than from the dropdown list, as illustrated in Figure 11.9. Once this is set, you can then select the specific increment from the dropdown below the Filter By: category. In this case the distance is 25′-0."

In many cases, you may wish to filter based on additional criteria. Note that each item must be in the field for it to display. As more items are added to this filter, you increase the risk of having absolutely nothing show up in the actual schedule. If this happens, back yourself out of some of these filters until you can see the data again. Notice in Figure 11.10 that there are no framing members that are shorter than 25′-0″.

FIGURE 11.9
The Filter function



You will notice that each item is independently listed. For some scheduled items, this may be exactly what you want to see. For others, such as a list of beams that exceed 25′-0″ you may want to sort each member that is identical. This can be accomplished within the Sorting/Grouping tab.

FIGURE 11.10
25´-0˝ minimum

Structural Framing Schedule		
Type	Cut Length	Count
W14X30	29' - 3"	1
W14X30	29' - 1 1/4"	1
W14X30	28' - 6 3/4"	1
W14X30	28' - 6 3/4"	1
W14X30	26' - 2"	1
W14X30	26' - 3 1/8"	1
W14X30	28' - 7 7/8"	1
W14X30	28' - 6 3/4"	1
W14X30	28' - 6 3/4"	1
W14X30	28' - 6 3/4"	1
W14X30	29' - 1 1/4"	1

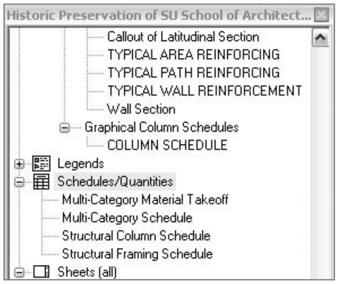
SORTING/GROUPING

The Sorting/Grouping tab of the Schedule Properties dialog will allow you to break down the data in an organized format. It is here where you can produce line-item counts and totals. Also, you can tell Revit to separate each grouping with its own line to report grand totals as well as an overall counting of items.

To get to the Sorting/Grouping tab, you can do one of the following:

- If you are not in a schedule at all, you can find the Schedule/Quantities category in the Project Browser. You will have to scroll down a little, but you will see it just above the sheets as illustrated in Figure 11.11. Once in the schedule, right-click and select View Properties. Now, select the Edit button in the Schedule Quantities row.
- If you are already viewing the schedule, you can right-click and select View Properties. From there click the Edit button in the Schedule Quantities row.

FIGURE 11.11
Finding the Schedules/Quantities in the Project Browser



In the Sorting/Grouping tab, the first thing you will see is the Sort By: field. There are multiple Sort By: fields. This allows you to create a sorting hierarchy. Normally, a schedule's first sorting criteria will be based on either the name or type. The second criteria will then be based on the differentiating item. In the case of the example illustrated in Figure 11.12, the framing members are sorted first by type, then by cut length.

For each group, you can add either a Header or a Footer. This helps in the organization of the data as it is displayed. If you choose to provide a Footer for each group, you can provide a title, counts and totals, or a combination. To the right of the Header/Footer menu is a toggle which allows you to add a blank line. This is sometimes checked to further break up the item listings. See Figure 11.12.

Towards the bottom of the dialog, you will see a category which allows you add grand totals. This will count every item in the schedule regardless of its sort criteria, giving you an overall total of the items being queried. Just below the Grand Totals checkbox is the control to Itemize Every Instance. If unchecked, this will group all identical items. See Figure 11.12.

Adding a Header or a Footer will provide you with the space necessary to add a general count and total. There will be times where you need these line items at the end of each column. The Formatting tab will allow you to do this. See Figure 11.13 for an example of a sorted schedule.

FIGURE 11.12
The Sorting/Grouping tab

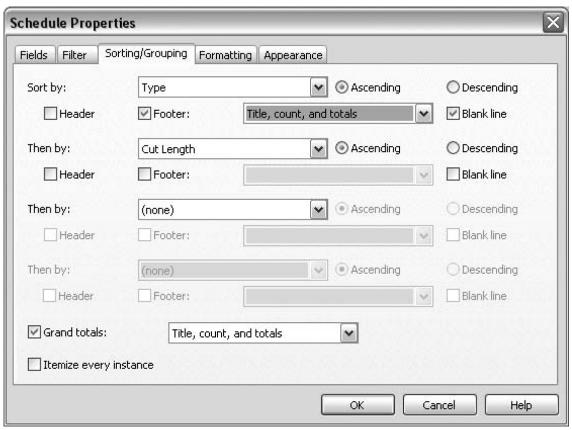


FIGURE 11.13
The sorted Schedule

Structural Framing Schedule		
Type	Cut Length	Count
16K7	27' - 8 3/8"	1
16K7	30' - 0"	10
16K7	30' - 0"	1
16K7: 12		
W12X26	26' - 8 1/8"	13
W12X26	29' - 1 1/4"	2
W12X26	29' - 2 1/4"	1
W12X26	29' - 3"	1
W12X26	29' - 3"	3
W12X26	29' - 4 1/4"	169
W12X26	29' - 5"	1
W12X26	29' - 7 1/2"	1
W12X26: 191		
W14X30	25' - 11 7/8"	13
W14X30	26' - 2"	14
W14X30	26' - 3 1/8"	14
W14X30	26' - 9 1/2"	1
W14X30	28' - 6 3/4"	56
W14X30	28' - 6 3/4"	14
W14X30	28' - 7 7/8"	14
W14X30	29' - 1 1/4"	98
W14X30	29' - 2 3/4"	39
W14X30	29' - 3"	98
W14X30	29' - 3 5/8"	13
W14X30	29' - 5"	1
W14X30	29' - 6 5/8"	3
W14X30	45' - 2 3/8"	14
W14X30: 392		
Grand total: 595		

FORMATTING

In the Formatting tab, you can specify the behavior of each field in the schedule. This gives you an additional advantage over how you are going to relay the information on the sheet. On the left side of the tab, you get a field which displays the fields that you have added to the schedule. On the right side you have the formatting choices:

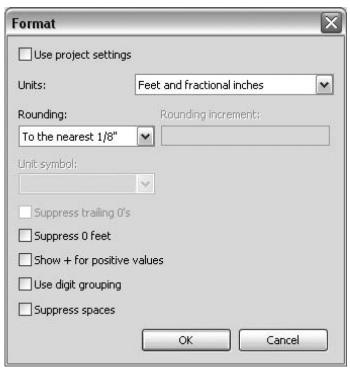
Heading A text field which allows you to label the column. Normally the only change here is to capitalize the text. But you can, of course, change the entire field to be labeled whatever you wish.

Heading Orientation Allows a rotation of either horizontal or vertical. This is normally kept horizontal; however, there will come a time where you will need this specific function.

Alignment Justifies the text based on the traditional left, center, right justification.

Field Formatting This category pertains mostly to numeric columns. You can click on the Field Format button to gain access to the Format dialog, as illustrated in Figure 11.14. If you want to format this field differently than the project's units are formatted, you can un-check the Use Project Settings control at the top of the dialog.

FIGURE 11.14
The Format dialog



The Calculate Totals checkbox will allow you to add a total into the Footer for the grouping's row. This information is provided on a row-by-row basis, and can only be displayed if it is checked on here, as illustrated in Figure 11.15.

If you do not want your column to be displayed at all, you can select the Hidden Field checkbox. This is provided here so that you can still sort and group your schedule based on the information in this field, but may not want to show it. Normally, this is unchecked, as illustrated in Figure 11.15.

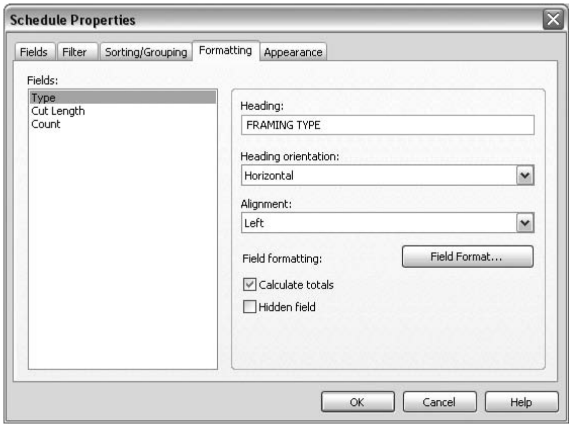


FIGURE 11.15 The Formatting tab

As you continue to modify items in the dialogs, you will probably need to bounce back and forth between the finished schedule and the view properties quite a bit until you finally see the information how you would like it displayed, as illustrated in Figure 11.16. You should remember, however, that this procedure does not have to be done every time you need a schedule. Once a good schedule is created, it should be added to your company's template.

FIGURE 11.16 The formatted Schedule

Structural Framing Schedule

FRAMING TYPE	CUT LENGTH	COUNT
16K7	27' - 8 3/8"	1
16K7	300' - 0"	10
16K7	30' - 0"	1
16K7: 12	357' - 8 3/8"	
W12X26	346' - 8 7/8"	13
W12X26	58' - 2 1/2"	2
W12X26	29' - 2 1/4"	1
W12X26	29' - 3"	1
W12X26	87' - 9 1/8"	3
W12X26	4961' - 1 1/2"	169
W12X26	29' - 5"	1
W12X26	29' - 7 1/2"	1
W12X26: 191	5571' - 3 7/8"	
W14X30	337' - 10 5/8"	13
W14X30	366' - 4 3/4"	14
W14X30	367' - 7 3/8"	14
W14X30	26' - 9 1/2"	1
W14X30	1599' - 7 1/8"	56
W14X30	399' - 11 1/8"	14
W14X30	401' - 3 1/8"	14
W14X30	2852' - 4 3/8"	98
W14X30	1140' - 0"	39
W14X30	2866' - 9 7/8"	98
W14X30	380' - 11 1/2"	13
W14X30	29' - 5"	1
W14X30	88' - 7 7/8"	3
W14X30	632' - 8 5/8"	14
W14X30: 392	11490' - 4 7/8"	
Grand total: 595	17419' - 5"	

This information is almost ready to be placed on a drawing sheet. If this schedule was created with the intention that it is for information only, without being placed on a sheet, then the next topic is unnecessary. If this schedule is going to be displayed, then the Appearance tab will need to be configured.

Appearance

The Appearance tab is broken down into two categories:

- Graphics
- Text

In the Graphics category, you are concentrating on the lines and the grid configuration. The first checkbox you see is a control for the display of the grid lines as illustrated in Figure 11.17. Normally, you are going to have the grid lines turned on. It gets confusing very quickly if you don't. This is true for a schedule which is for information only as well as a schedule that is to be placed onto a sheet. Typically, the grid lines are going to show up very lightly. You have the control over the line weight in the menu as shown. To the right of the grid line and the Line Weight menu there is a checkbox for the grid in Headers/Footers/Spacers. This control pertains to the schedule as it appears on the sheets. The default for this is to be checked on.

The next checkbox pertains to the Outlines. This is for the display of the schedule as it is placed on a sheet. Typically there is an outline. You can change the line weight as you please, as illustrated in Figure 11.17.

The Blank Row Before Data checkbox is added to serve as a separator between the headers and the data. This can be turned on.

FIGURE 11.17 The Graphics category

In the Text category, you are going to configure the appearance of, obviously, the text. The first checkbox you will see is the Show Title control. This allows you to place an overall title on the schedule. You can check the Underline toggle which will add a grid line under the entire title. Of course you can then thicken the line to your company's standards by selecting a line weight from the menu, as illustrated in Figure 11.18.

The next item you will come to is the Show Headers checkbox. This is normally checked, as illustrated in Figure 11.18.

Underneath the Show Headers toggle are the formatting options for the header and the body text. This allows you to change the font and the font size. You can also bold and italicize the text as you see fit, as illustrated in Figure 11.18

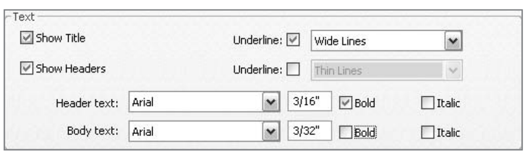


FIGURE 11.18 Formatting the text

Now that you have explored the five tabs in the Schedule/Quantities dialog, it is time to go back and look and at how you can start to add formulas to fields in your schedule that can give you additional functionality.

Calculated values

When adding a calculated value, you are basically adding a mathematical formula to your schedule, similar to a cell in a spreadsheet. The big difference is, you are not calculating a cell intersection such as *G2*A4*. In Revit the formula is textual, and would read like: *Material: Volume * 2*. This would be the volume of whatever material you are scheduling multiplied by 2. Of course an "out of the box" schedule will only get you so far in terms of the information you want reported back to you, so these calculated values need to be added by you. If these are common equations, you should definitely have this calculated value in a pre-created template so the end user does not have to create them over and over again.

A good example of using a calculated value would be in a floor schedule where the designer needs a total floor volume based in cubic yards, with a given waste factor.

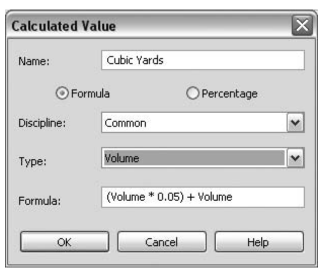
To do this, create a new schedule following the steps listed below:

1. Go to View ➤ New ➤ Schedule/Quantities.
2. In the Schedule/Quantities dialog, select Floors.
3. Add the fields:
    - Level
    - Area
    - Volume
4. In the Fields dialog, you will see a Calculated Value button between the Available Fields and the Schedule Fields columns, as shown in Figure 11.19. Click this button.

FIGURE 11.19
Adding a calculated value

5. In the Calculated Value dialog illustrated in Figure 11.20, fill in the name as Cubic Yards. The discipline is Common, and the type is Volume.
6. In the Formula field, enter: *(Volume * 0.05) + Volume*. This will take the Volume field and multiply it by a factor of .05. This result is then added to the volume again creating the volume plus the waste. See Figure 11.20.

FIGURE 11.20
The Calculated Value dialog



7. Once the field is created, click on the Formatting tab, select the newly created Cubic Yards field, and click on the Field Format button.
8. In the Field Format dialog, uncheck Use Project Settings and set the Units to Cubic Yards, as illustrated in Figure 11.21.

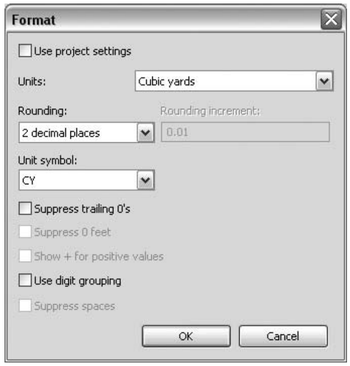


FIGURE 11.21
Formatting the new field

9. Click OK twice to get back to the schedule view. You will see the new field, as illustrated in Figure 11.22.

FIGURE 11.22
The completed schedule

Floor Schedule			
Level	Area	Volume	Cubic Yards
1ST FLOOR	15582 SF	6492.45 CF	252.48 CY
2ND FLOOR	15582 SF	6492.45 CF	252.48 CY
3RD FLOOR	15582 SF	6492.45 CF	252.48 CY
4TH FLOOR	15582 SF	6492.45 CF	252.48 CY
5TH FLOOR	15582 SF	6492.45 CF	252.48 CY
6TH FLOOR	15582 SF	6492.45 CF	252.48 CY
7TH FLOOR	15582 SF	6492.45 CF	252.48 CY
8TH FLOOR	15582 SF	6492.45 CF	252.48 CY
9TH FLOOR	15582 SF	6492.45 CF	252.48 CY
10TH FLOOR	15582 SF	6492.45 CF	252.48 CY
11TH FLOOR	15582 SF	6492.45 CF	252.48 CY
12TH FLOOR	15582 SF	6492.45 CF	252.48 CY
13TH FLOOR	15582 SF	6492.45 CF	252.48 CY

Schedule Keys

Creating a schedule key is a slightly different approach than creating the schedule you just did. The intent of a schedule key is to simply have a key designation, then a simple description of that designation. For example, if you have many different sized footings you need to keep track of, you can add them to a schedule key, and tie that key into the actual footing type.

Creating a schedule key is the same process as creating a regular schedule. First go to View ➢ New ➢ Schedules/Quantities. Once you select an item you wish to schedule from the right hand column, you can pick the Schedule Key button, as illustrated in Figure 11.23.

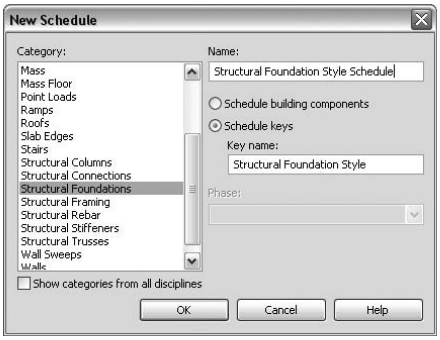


FIGURE 11.23
Selecting a schedule key

In the Fields tab, you will see you have only two items to add to the schedule. Make sure both the Key Name and the Comments fields are both added, and click OK. This will bring you to a seemingly blank schedule. At this point, you can right-click and select New Row, as illustrated in Figure 11.24

FIGURE 11.24
Adding a new row to the key

Give the new row a key name of whatever you wish (F1 is applicable for this example), and give it a description based on the item (Rectangular footing 36x36 is applicable here).

Now, at this point you can click OK to exit this schedule. You can find the actual component that is being scheduled. In this example, specific rectangular footings are being scheduled. Once the item(s) are selected, click the Element Properties button from the Options bar. In the Element Properties dialog, you will see a field called Structural Foundation Style under the Identity Data heading. By default this will be set to (None). If you click the drop list arrow, you will see F1 available; select it. The Comment field will then change to reflect the comment you added to the schedule. See Figure 11.25. As you can see, creating a schedule in Revit is not a difficult task, and you can display quite a bit of information from the model. After the following exercise, you will take a step further and start pulling material quantities from the model.

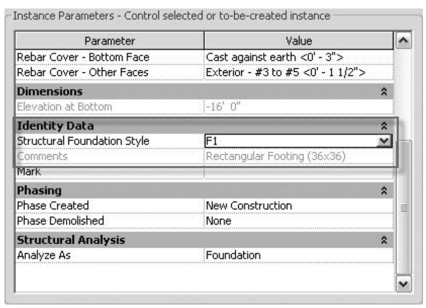


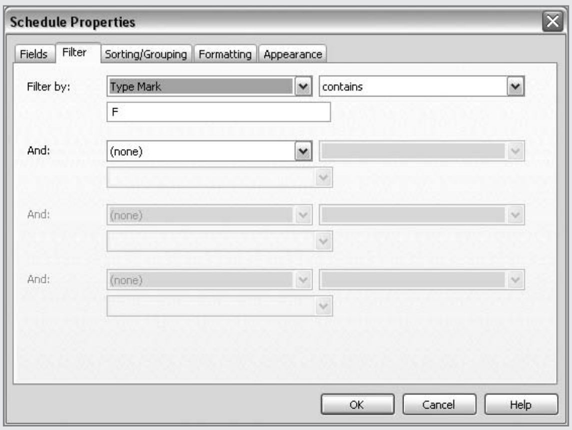
FIGURE 11.25
Adding the key to the objects

Exercise: Creating a schedule from scratch

The following exercise will guide you through the steps involved in creating a footing schedule. The example used here can then be applied to your company's specific needs.

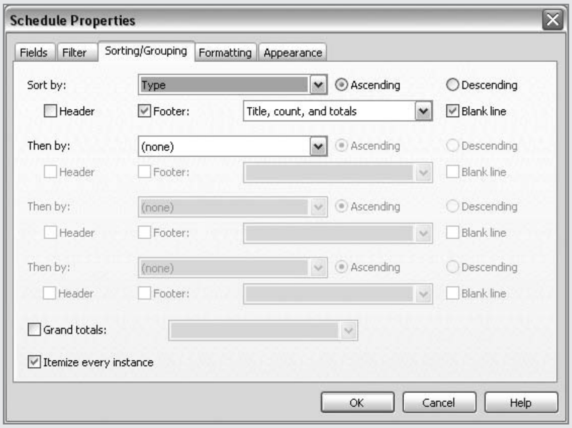
To begin, open the file called `schedules.rvt`, provided on our website, and follow these steps.

1. With the `schedules.rvt` file open, activate the T.O. FOOTING plan view. There you will see, of course, the footings. Notice that there are two different kinds.
2. Go to View ➢ New ➢ Schedule/Quantities.
3. In the New Schedule dialog, select Structural Foundations. You can leave the other settings at their defaults.
4. Click OK.
5. In the Fields tab (it should be the default tab), you want to start adding fields to the schedule. In the left column (Available Fields:) scroll down and select Type.
6. Between the columns there is an Add button. Click it. The Type field moves to the right. It is now officially part of the schedule.
7. Do the same for the Family field, and Type Mark.
8. Click on the Filter tab.
9. Filter by Type Mark.
10. In the field to the right, select Contains from the drop list.
11. In the edit box below the Filter By: category type the letter **F**.

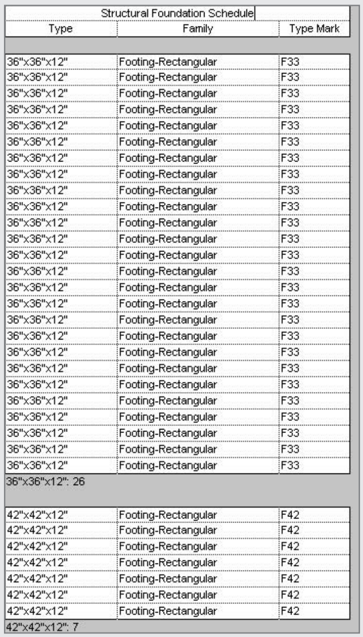


12. Click to the Sorting/Grouping tab.
13. Use the Sort By: drop list and choose Type.

14. Turn on the Footer checkbox and then choose Title, Count and Totals. Also turn on the Blank Line checkbox.



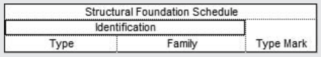
15. Click OK. Your schedule is now shown in two groups. Other foundation systems that are not tagged with a type mark that at least contains the letter F are excluded from this schedule.



Structural Foundation Schedule		
Type	Family	Type Mark
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12"	Footing-Rectangular	F33
36"x36"x12": 26		
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12"	Footing-Rectangular	F42
42"x42"x12": 7		

16. At the top of the schedule you will see the headings Type and Family. Click and hold on the Family cell, and drag your mouse to the left, selecting both cells.

17. On the Options bar, you will see a button that says Group. Select it. The headers are now grouped together.
18. Name the new cell **Identification**.



Now that the schedule is created, you should not waste your efforts. Save this little gem to your standards.

Now for the quantity part of Revit. This is where the 4D part that you have been hearing so much about over the last few years comes from. With the same procedure that you just applied to the schedules, you can practice that knowledge to produce a material takeoff.

Material Takeoff Schedules

A nice thing about Revit Structure that you will find is that many of the processes are similar, if not exactly the same to one another. Creating a material take-off, compared to creating a schedule, falls under this category. Once you have mastered one method, the other will fall right into place. By going to the View menu, you can then go to New ➢ Material Takeoff. A good example for this would be to find a concrete structure with pan formed joists. The material that comprises the concrete framing can be quickly quantified and totaled. Within the New Material Takeoff dialog, you can select Structural Framing, and click OK, as shown in Figure 11.26.

If you are following along with this chapter, you can open the `Material Takeoff.rvt` file found on the website.

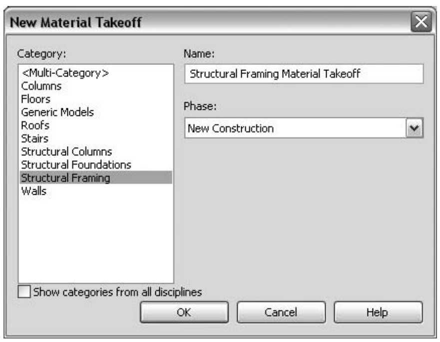


FIGURE 11.26
Creating a new material takeoff

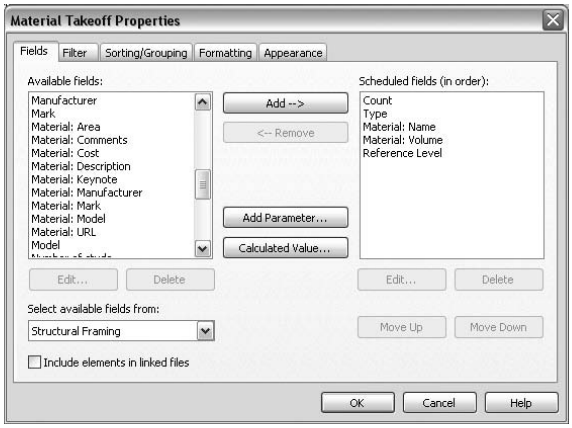
As you start adding fields in the next dialog you can scroll down in the Available Fields column, and you will notice some new types of fields that have the prefix of Material:. You can then add fields such as:

- Count
- Type

- Material: Name
- Material: Volume
- Material: Level

See Figure 11.27 for an illustration.

FIGURE 11.27
Adding materials to the schedule

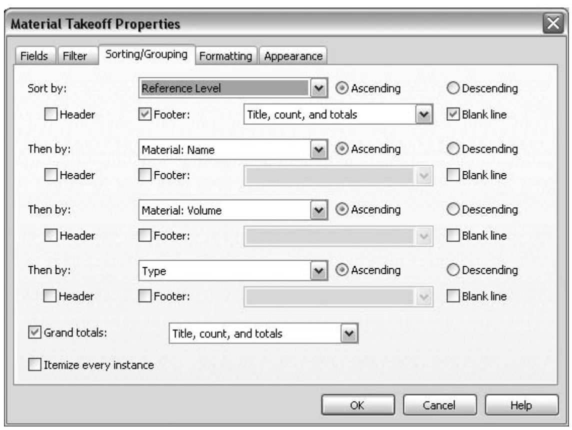


This type of schedule is sometimes heavily dependent upon how the fields are sorted. If you go to the Sorting/Grouping tab, you can sort the fields up to (4) different categories. Also, it is important to create footers, and to total the values. At the bottom you can also uncheck Itemize Every Instance. In this example, the fields are sorted by:

- Reference Level
- Material: Name
- Material: Volume
- Material: Type

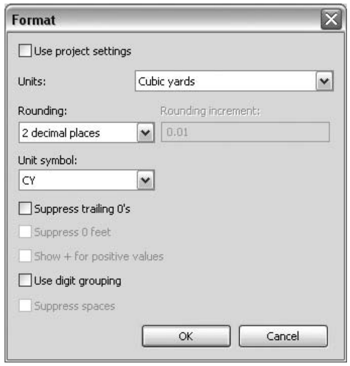
See Figure 11.28 for an illustration.

FIGURE 11.28
Sorting the material takeoff



An important field in this example is the Material: Volume field. For an accurate material takeoff, this field can be formatted and calculated independently of the rest of the schedule. By going to the Formatting tab, you can select the Material: Volume field, and for Field formatting, check the Calculate Totals checkbox. Also, select the Field Format button. This will allow you to override the default formatting of this field. You can now select Cubic Yards for the formatting of this field as illustrated in Figure 11.29.

FIGURE 11.29
Field formatting



You now have an exact material takeoff for the volume of concrete needed to cast the framing for this structure, listed in cubic yards. As the design of the building changes, so do the takeoff values, resulting in an accurate on-the-spot quantification of the concrete volumes throughout the life of the project.

Another good example is to perform a floor area calculation. This takeoff can contain both area and volume information serving as a dual purpose schedule/takeoff. To follow along, you can go to View ➢ New ➢ Material Takeoff. This time select Floors from the Category: list. For the fields, add:

- Material: Name
- Material: Area
- Material: Volume
- Level

In the Sorting/Grouping tab, you can sort the materials any way you choose. This example will be sorted by Material: Name, then by Level. Footers are turned on for both categories.

In the Formatting tab, the Material: Area field is selected, and the Calculate totals checkbox is on. For Material: Volume, the Calculate totals checkbox is on, and the Field Format is overridden to reflect Cubic Yards. See Figure 11.30.

The result is another schedule/takeoff that is accurate and can be produced at any stage in the life of the project. See Figure 11.31. This schedule can be used in a multitude of ways. One way is to relay this information on the construction documents by adding the schedule to a drawing sheet, another is to simply use it as a material takeoff, and a third is to actually use the schedule to drive design changes back to the model. This third method will be explored next.

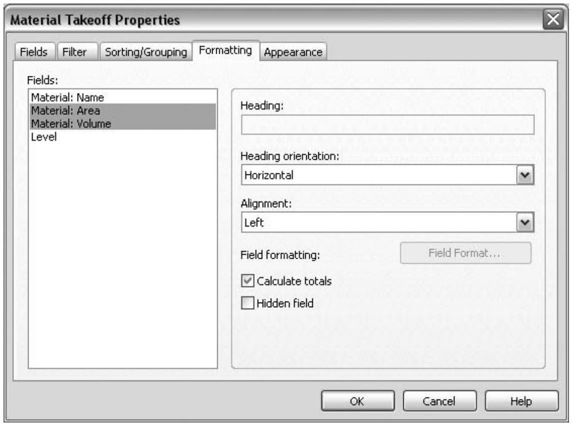
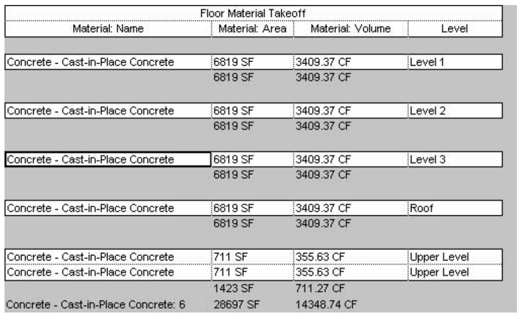


FIGURE 11.30
Formatting the floor area material takeoff



Floor Material Takeoff

Material: Name	Material: Area	Material: Volume	Level
Concrete - Cast-in-Place Concrete	6819 SF	3409.37 CF	Level 1
	6819 SF	3409.37 CF	
Concrete - Cast-in-Place Concrete	6819 SF	3409.37 CF	Level 2
	6819 SF	3409.37 CF	
Concrete - Cast-in-Place Concrete	6819 SF	3409.37 CF	Level 3
	6819 SF	3409.37 CF	
Concrete - Cast-in-Place Concrete	6819 SF	3409.37 CF	Roof
	6819 SF	3409.37 CF	
Concrete - Cast-in-Place Concrete	711 SF	355.63 CF	Upper Level
Concrete - Cast-in-Place Concrete	711 SF	355.63 CF	Upper Level
	1423 SF	711.27 CF	
Concrete - Cast-in-Place Concrete: 6	28697 SF	14348.74 CF	

FIGURE 11.31
Another completed schedule

Editing Schedules

As you have seen, any change in the model will have an influence on the schedule. But suppose you wanted the opposite to happen? It sure would be nice if you could change an item in the schedule and have it propagate back to the model physically. Well, in Revit, you can do just that. It makes sense if you think about it. If the schedules are always an accurate representation of the model, it seems like it should be impossible to change a schedule item and not have it affect the model. Well, it is impossible. If you change an item in a schedule it will change the model.

Not every field in a schedule is modifiable. If the field in the schedule is a calculated result (such as volume) then the field cannot be changed in the schedule. It must physically be changed in the model. If the field is a parameter that be modified in the Objects properties (such as Family Type) then it can be modified in the schedule. For example, Figure 11.32 shows a row of fields. The first field, Count, is a calculated value. Therefore it cannot be changed here unless an identical instance of this item is created in the model. The second field, Type, can be modified in the schedule.

You will be able to tell which items are able to be modified by clicking into the cell. If a menu appears with other selections, you can choose one of these other choices, and the schedule and the model will be updated to reflect this change. The fields Top Level and Base Level can also be changed in the schedule; however Volume cannot.

Figure 11.32
Displaying fields that can be modified

Structural Column Schedule				
Count	Type	Top Level	Base Level	Volume
1	24 x 24	Roof	T.O. Footing	5.74 CY
1	12 x 18	Upper Level	T.O. Footing	7.63 CY
1	18 x 24	Upper Level	T.O. Footing	6.59 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 30	Upper Level	T.O. Footing	7.37 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Roof	T.O. Footing	5.93 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Roof	T.O. Footing	5.93 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Upper Level	T.O. Footing	7.37 CY
1	24 x 24	Upper Level	T.O. Footing	6.59 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Upper Level	T.O. Footing	7.37 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Roof	T.O. Footing	5.93 CY
1	24 x 24	Roof	T.O. Footing	5.19 CY
1	24 x 24	Roof	T.O. Footing	5.93 CY
1	24 x 24	Roof	T.O. Footing	6.30 CY
1	24 x 24	Roof	T.O. Footing	6.30 CY
1	24 x 24	Roof	T.O. Footing	6.30 CY
1	24 x 24	Roof	T.O. Footing	6.30 CY
1	24 x 24	Upper Level	Roof	1.41 CY
1	24 x 24	Upper Level	Roof	1.41 CY

Real World Scenario

Proceed with caution!

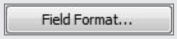
Being able to modify these schedules in this manner is a great thing, but it could also get you into trouble. Be sure that when you are altering information in a schedule you are doing it deliberately, and are checking the model after the changes have been made. Although Revit changes your approach to modeling, it does not negate the need for a thorough back-check of the documents.

Exercise: Creating a "working" schedule

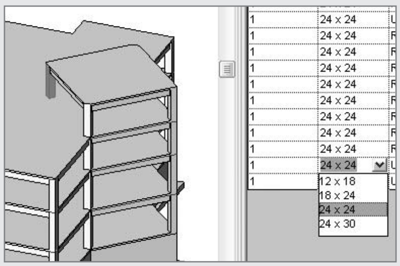
To practice creating a schedule and using it to modify the model, follow these steps:

1. Open the file called material `takeoff.rvt`.
2. Go to View ➢ New ➢ Schedule/Quantities.
3. Select Structural Columns.
4. Click OK.
5. Add the following fields:
   - Count
   - Type
   - Base Level
   - Top Level
   - Volume

6. Go to the Sorting/Grouping tab
7. Sort by Type
8. Go to the Formatting tab
9. Select the Volume field
10. Click the Field Format button



11. Uncheck Use Project Settings
12. Set the Units to Cubic Yards
13. Click OK
14. Check Calculate Totals
15. Click OK
16. In the model, select the Window menu, and select Tile. This will allow you to see both the 3D model and the schedule. This makes it easier to modify elements from the schedule because you can see the modification happen in real-time. This takes the guess work out of the process.
17. At the bottom of the schedule, there are two columns that run from the Roof to the Upper Level. These columns are too big. We can size them down.



18. Change the Type to 18 x 24. Notice the column changes in size, and the row in the schedule moves to the top.
19. Repeat the process for the other column that runs from the Roof to the Upper Level.

Exporting Schedules to a Spreadsheet

Although Revit Structure is a full, comprehensive database which is fully capable of performing most tasks with ease, there is still, and always will be, a need to export data to other formats. The topic of exporting the model to other CAD applications has been covered, but what about the data itself? Let's face it, you are going to find that not everybody is going to be using Revit. Actu-

ally, you are probably finding that most people aren't using Revit. This could even be true inside your office. An estimator or project manager probably will not be using Revit, but will still need to manually manipulate the data that you are outputting from the model.

To begin, you first need to create a schedule or a material takeoff in Revit Structure. In this example we are going to use a Walls material takeoff. This material takeoff consists of the fields:

- Count
- Material: Name
- Length
- Material: Area
- Material: Volume

The fields are Sorted upon the Material: Volume field along with a Blank line. This is done in the Sorting/Grouping tab. The parameters chosen in this example are:

- Grand totals of Title.
- Count and Totals is checked on.
- Itemize Every Instance is checked off.
- In the Formatting tab, the Material: Volume field's field format is overridden to Cubic Yards.
- Calculate Totals is also checked on for both Material: Volume and Area.

See Figure 11.34.

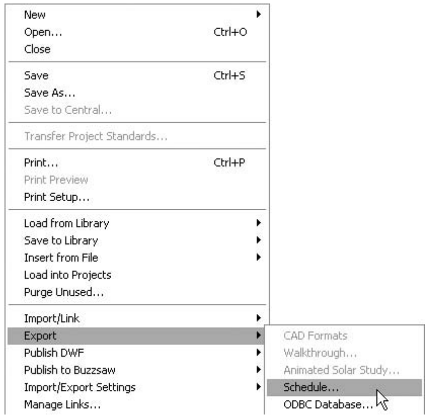
FIGURE 11.33
The Material Takeoff that will be exported to Excel

Wall Material Takeoff

Count	Material: Name	Length	Material: Area	Material: Volume
4	Concrete - Cast-in-Place Concrete	8' - 0"	320 SF	11.85 CY
1	Concrete - Cast-in-Place Concrete	23' - 6"	235 SF	8.70 CY
1	Concrete - Cast-in-Place Concrete	25' - 0"	240 SF	8.89 CY
4	Concrete - Cast-in-Place Concrete	25' - 0"	1012 SF	36.73 CY
4	Concrete - Cast-in-Place Concrete	25' - 0"	1000 SF	37.04 CY
3	Concrete - Cast-in-Place Concrete	25' - 0"	771 SF	28.01 CY
1	Concrete - Cast-in-Place Concrete	25' - 0"	255 SF	9.44 CY
Grand total: 18			3833 SF	140.66 CY

It's now time to export the material takeoff. One of the more not-so-obvious items you need to remember when exporting takeoffs and schedules is that you need to currently have the schedule you wish to export open. In many cases, individuals get the impression that they cannot export a schedule based on the fact that it is sometimes grayed out in the list. If the actual view is open, the choice to export a schedule is active. To get to this active export choice, be sure the schedule is actively displayed, and go to File ➢ Export ➢ Schedule, as shown in Figure 11.34.

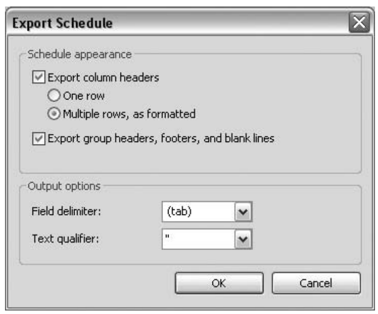
FIGURE 11.34
Finding the active export function



The next dialog will be a Export Schedule file Save As dialog. Notice the only data type is a delimited .txt file. This is fine because most databases and spreadsheet programs know how to read these types of files. Once you find the directory into which you wish to place the .txt file, click Save. It is recommended that you have a folder set up in the project's job directory where all of the team members have access to the files being generated from Revit. Also, it is a great idea to have meetings and conversations alerting the team to expect this kind of information and exactly where it is going to be stored.

The Schedule appearance dialog will allow you to control how the data is going to be transferred. The defaults for the schedule's appearance are to export column headers, and to create multiple rows as formatted in Revit. It is recommended that you keep these defaults as it can get confusing in Microsoft Excel trying to sort out this information manually. Also, this will probably be viewed by a person who needs the headers to see where they are in the spreadsheet. Another schedule appearance default is to Export As Multiple Rows, as formatted in Revit. Again, it is recommended that you keep this setting for organization of the spreadsheet. The defaults for the Output options are to delimitate the fields based on (tab) type spacing. Traditionally this data exchange has been comma delimited, but a tab is a much cleaner import from the Excel side. The Text qualifier is, of course, in quotations. See Figure 11.35.

FIGURE 11.35
Setting the exported data options



Once the .txt file has been made, it can now be imported into a spreadsheet. This example uses Microsoft Excel 2007 for the data exchange. To import the data, open Microsoft Excel. Once in Excel, select Open. For the files of type, select Text files (*.prn; *.txt; *.csv) as shown in Figure 11.36. Browse to the directory where your .txt file is stored, and select the file.

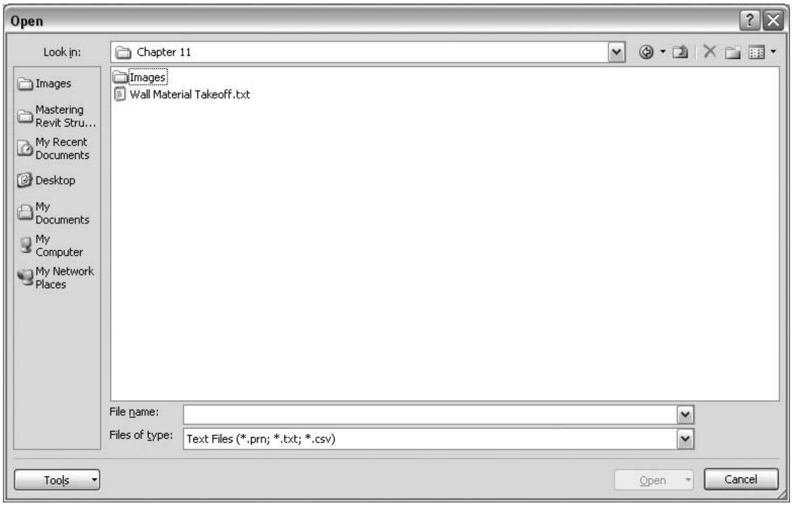


FIGURE 11.36
Opening a .txt file in Microsoft Excel.

Excel will now prompt you to make some additional choices as shown in Figure 11.37, the Text Import Wizard. Essentially, this is a confirmation of the dialog shown in Figure 11.35. We are still using tab delimitated data. The file origin is coming from OEM United States. You can also have a very basic preview of the data in the bottom window of Figure 11.37.

FIGURE 11.37
The Text Import Wizard in Microsoft Excel

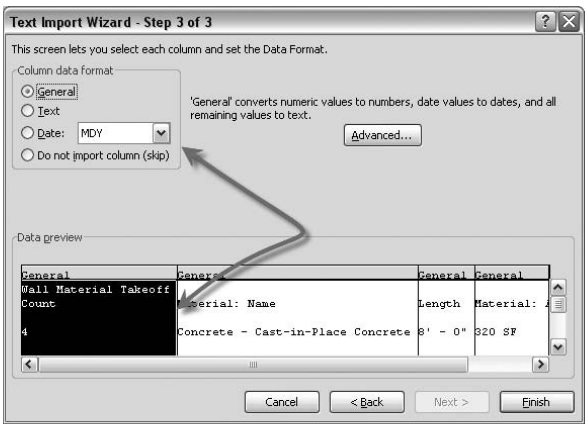
Since this is a wizard, you will see a Next button at the bottom of the dialog. Clicking this will bring you to the next step in the process as illustrated in Figure 11.38. Again, the defaults are simply a reflection of the initial setup you did when exporting the data from Revit. The Delimiters

field is set to Tab Delimited. You can add additional delimiters, but this will probably adjust the data to an unreadable format once it is brought into Excel. On the contrary, however, if the data comes in unreadable or un-organized, you may have to go back and make some adjustments to these fields until you have the data the way you wish to see it. Click Next to continue.

FIGURE 11.38
Step 2 in the Text Import Wizard

The third step in the Text Import Wizard is the formatting of the data in the columns as illustrated in Figure 11.39. At the top of the dialog you have a field which allows you to select the type of formatting as applied to the row that is selected in the Data Preview underneath. Notice the Data Preview is a selectable table. You can format the entire column by first selecting a column here, and then make the formatting choice in the Column Data Format field above, as illustrated in Figure 11.39. Click Finish.

FIGURE 11.39
Step 3 in the Text Import Wizard



Once you have clicked Finish, the Excel table will appear. All of the data has been imported. Of course some of the columns may need to be adjusted to view the data better, and perhaps even some fill could be applied to the rows to delineate the fields as shown in Figure 11.40. Notice at the bottom of the table, the tab is labeled as the View Name in Revit. This can come in handy if

you have another Excel file set up with calculations that can be linked to this tab. Of course if you are doing this, be careful.

FIGURE 11.40
The final Excel spreadsheet

	A	B	C	D	E	F
1	Wall Material Takeoff					
2	Count	Material: Name	Length	Material: Area	Material: Volume	
3						
4	4	Concrete - Cast-in-Place Concrete	8' - 0"	320 SF	11.85 CY	
5						
6	1	Concrete - Cast-in-Place Concrete	23' - 6"	235 SF	8.70 CY	
7						
8	1	Concrete - Cast-in-Place Concrete	25' - 0"	240 SF	8.89 CY	
9						
10	4	Concrete - Cast-in-Place Concrete	25' - 0"	1012 SF	36.73 CY	
11						
12	4	Concrete - Cast-in-Place Concrete	25' - 0"	1000 SF	37.04 CY	
13						
14	3	Concrete - Cast-in-Place Concrete	25' - 0"	771 SF	28.01 CY	
15						
16	1	Concrete - Cast-in-Place Concrete	25' - 0"	255 SF	9.44 CY	
17	Grand total: 18			3833 SF	140.66 CY	

EXERCISE: EXPORTING A SCHEDULE FROM REVIT TO MICROSOFT EXCEL

This will be a step-by-step example of how to create a wall material takeoff and export it to Excel.

1. Open the file called material `takeoff.rvt`.
2. Go to View ➢ New ➢ Material Takeoff.
3. Select Walls from the list to the left.
4. Add the following fields:
   - Count
   - Material: Name
   - Length
   - Material: Area
   - Material: Volume
5. Go to the Sorting/Grouping tab.
6. Sort by Material: Volume.
7. Do not include a Footer, but check Blank Line.
8. At the bottom of the dialog, check on Grand Totals, and select Title, Count and Totals from the list.
9. Uncheck Itemize Every Instance.
10. Go to the Formatting tab.
11. Select Material: Volume and click the Field Format button.
12. Uncheck the Use Project Settings button, and change the formatting to Cubic Yards.
13. Click OK.

14. Click on Calculate Totals.
15. Click OK.
16. Click OK again.
17. Go to File ➢ Export ➢ Schedule.
18. Browse to the directory into which you wish to save the .txt file.
19. Click Save.
20. In the Export Schedules dialog, keep the defaults.
21. Click OK.
22. Open Microsoft Excel.
23. Open the .txt file.
24. In the first of three steps of the Text Import Wizard, keep the defaults to be Delimited.
25. Click Next.
26. In the second step of the Text Import Wizard, keep the delimiters to be Tab.
27. Click Next.
28. In the third step of the Text Import Wizard, keep the Column Data Format to be general.
29. Click Finish.
30. In Excel, spread the cells apart so you can see the data in the header rows.
31. Notice the bottom tab is labeled the same as the material takeoff from Revit.
32. Click the Save button in Excel.
33. Save the file as a Microsoft Excel (.xlsx, or .xls) file type.

This procedure is quite useful for getting your data safely over to the estimator, or just in the interest of maintaining a "conventional" workflow between the drawings and the estimate/material takeoff. Keep practicing this routine, and before long you will have a go-to standard you can rely on.

Keynote Legends

One of the best ways to track items in a model is to use good, old-fashioned keynotes. One thing about Revit Structure, however, is that nothing is keynoted out-of-the-box. Another thing about Revit Structure is that you must first tag a keynoted item for it to show up in a legend. In Revit Structure, keynoting an item is a two step process. The nice thing is, once you have the item keynoted in your template, you won't have to deal with that specific item again!

To create a new keynote legend, you can go to View ➢ New ➢ Keynote Legend, as shown in Figure 11.41.

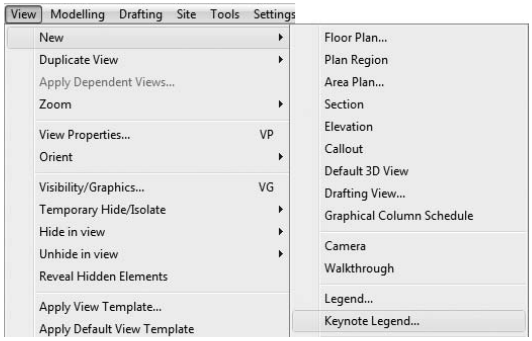


FIGURE 11.41 Creating a new keynote legend.

Before you can begin creating a legend, you must give it a name. In this example, the legend will keep track of foundation items, so the keynote legend will be called Foundation Keynote Legend.

A nice thing about creating keynote legends is you have two choices of fields to add to the legend. You can choose either Key Value, or Keynote Text (See Figure 11.42).

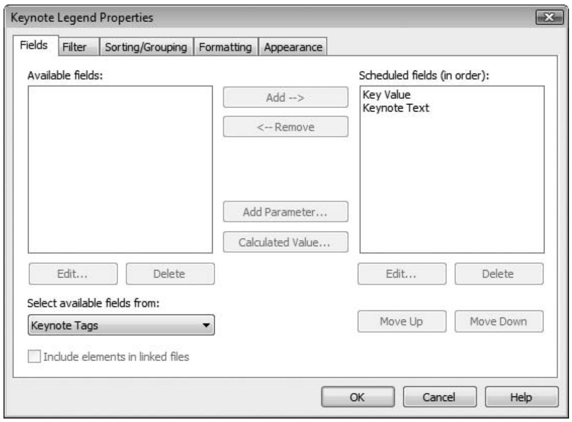


FIGURE 11.42 Adding fields to a keynote legend is quite simple.

Once the fields are automatically added, and you click OK, you will get a blank Keynote Legend. It's blank because, as mentioned before, you need to assign a key value to an item. As you have been learning in this chapter, you can modify an item by either accessing the Element Properties of that item, or make modifications directly from the schedule itself. Keynoting is no different!

To add a keynote value to a "blank" keynote legend, simply click in the first blank row. Once you see the […] button, you can click it. This will bring up the Keynote list as illustrated in Figure 11.43.

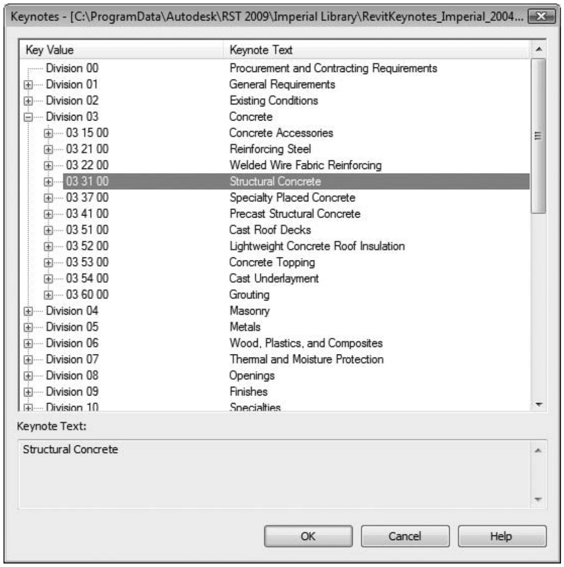


FIGURE 11.43
Adding a key value.

For this example, Structural Concrete is selected. Once you pick OK in the keynote list, the new keynote will be added to the legend.

Adding a Keynote Tag

You can add a tag with or without an accompanying legend value. If you do have a legend value (Structural Concrete) the tag will be placed when you pick the item associated with that keynote. If the item you wish to keynote does not have an accompanying keynote value, you will be promoted with the list upon placement of the tag.

To place a keynote tag, you can go to the Drafting tab on the Design bar. You will then see a Keynote button. If you pick it, you can choose Element, as shown in Figure 11.44.

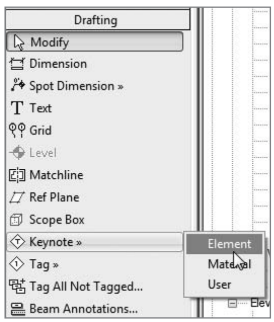
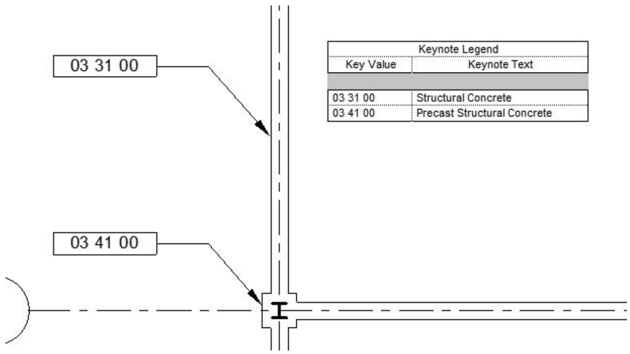


FIGURE 11.44
Adding a keynote tag to an element.

If you pick an item that happens to have the key value, you can place the tag. By default, the tag will be placed with the keynote number within a box. You can change this by simply choosing the desired keynote tag from the Type Selector on the Options bar as you are placing the tag. A popular alternative from the default is to place the Keynote Text.

If you pick an item that does not have an associated keynote value, you will get the list to choose from. In this example, two items are being keynoted. One is the structural foundation wall which will receive the Structural Concrete keynote. On this item, the tag is already filled out with the keynote information. The other is a precast column which will bring up the keynote list when placed because there is no keynote associated with it yet. Once you pick a new keynote for this item from the list, the tag will be filled out. In addition to the tag being filled out, an new line item will be added to the keynote legend itself. (See Figure 11.45.)

FIGURE 11.45
The Keynote Legend is growing.



The Bottom Line

Create Schedules Revit Structure has a good strong link between schedules and data. Once a schedule is created, the information can be manipulated either in the model or the schedule. Each will influence the next.

> **Master it** Building a schedule can start in one of two places. One of the places involves using the View tab of the Design Bar, and selecting the Schedule/Quantities button. What is the second way of starting a schedule?

Create Material Takeoffs Material takeoffs are virtually the definition of BIM. Revit Structure can really become an advantage to project costing in a real-time sequence. Once the tools are developed, and the basic takeoffs are in place, you can really start to see the benefits of taking the time to learn this feature.

> **Master it** Creating a material takeoff is different than creating a schedule; however, it is similar in many ways. What is the major difference between creating a material takeoff and creating a schedule?

Create Legend Schedules Legend schedules are a great way to sequence items in Revit. The nice part about this functionality is that you can simply draft items as well. This gives the user much more flexibility as they create legends for their model.

Master it A legend schedule is tied directly into the item being scheduled. What is the procedure for adding a legend schedule number to an item?

Send Schedules to Microsoft Excel Keeping track of the quantities may not be done by the designer. Many times it is an estimator who is not involved with the modeling process at all. This functionality allows the designer to output accurate data to an estimator in a format they are used to.

Master it Once a schedule or material takeoff is created in Revit Structure, how is it exported to Excel?

Chapter 12

Sheets

One aspect of using Revit Structure for BIM is creating documentation to help display the information that is part of the Revit Structure model. The method for creating these documentation pieces is pretty much the same as it is in a 2D environment. You create sheets with titleblocks and then add plan views and sections or details to the sheets along with schedules and notes for plotting. In addition, you may include a sheet index and perhaps a few key plans if required. Since all this information is stored in one database, Revit Structure can help keep track of everything as the specs change during the coordination process.

You can easily keep track of these changes for others to see by using Revit Structure's built-in revision tracking. As changes are made and tagged in views, revision schedules placed in titleblocks are automatically updated.

In this chapter you will learn to:

- Create a titleblock to display project information
- Create a revision schedule to your company standards
- Explore the behavior of the various view types when they are placed on a sheet
- Produce a sheet index to keep track of your issued sheets
- Control the behavior of revisions in your project

Getting to Know Your Sheets

Sheets are a category in the Project Browser for organizing a system family called Drawing Sheets. Revit Structure refers to these drawing sheets as simply *sheets*. Inside this system family you place other views and external families such as titleblocks. Sheets are another type of view used to display information from the model. They are somewhat like a drafting view in that they do not display information directly from the model or have a view range, but they are much more powerful in managing the information that is put inside them.

Similarly to any other views, you can place line work and annotation directly into sheets. In order to display model information, you can place other views that display the model elements inside sheets. Once other views are placed within a sheet, Revit Structure keeps track of their references and placement on the sheet as well as manages the various scales that are required to show the various levels of detail in a plan or detail view. Since sheets don't have a scale of their own and they manage the scales of those views that get placed within them, it is no longer necessary to worry about scale factors, text height, and dimension settings. As these views get moved to different locations on a sheet or onto another sheet, or when sheet numbers change, Revit Structure automatically updates the references to them throughout the model.

The basic procedure to create a new sheet in your project is as follows:

1. Right-click the Sheets category in the Project Browser and select New Sheet, or choose View ➢ New ➢ Sheet.
2. Within the Select A Titleblock dialog shown in Figure 12.1, select a titleblock to use for the sheet. If none is available, click the Load button and browse to a titleblock family file to load.
3. After choosing a titleblock, click OK to create the new sheet.

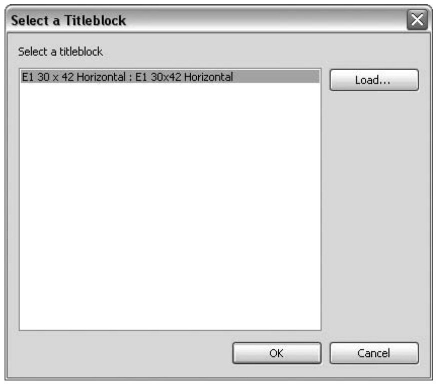


FIGURE 12.1
Select the titleblock to be used when creating new sheets.

RAPID-FIRE SHEET CREATION

So you want to create multiple sheets quickly rather than creating them individually by right-clicking and selecting a titleblock or by choosing View ➢ New ➢ Sheet. You can speed up this process a lot. In the Project Browser select a sheet and press Ctrl+C to copy it, or right-click the sheet and choose Copy to Clipboard. Activate the current sheet by clicking inside the view, and then press Ctrl+V to paste a copy of the sheet. A new sheet view will be created, numbered, and named, and its titleblock will be based on the previous sheet. You can create new sheets as fast as you can press Ctrl+V on the keyboard. If you have a tough time making this work, keep in mind that it works only on drawing sheets that have *no* views on them.

When a sheet view is created, its name is automatically associated with the sheet number and sheet name, which are also properties of the view. You can change this name by changing the parameter values in the properties of the view, by right-clicking the sheet view and selecting Rename or by selecting the sheet and pressing the F2 key, as shown in Figure 12.2.

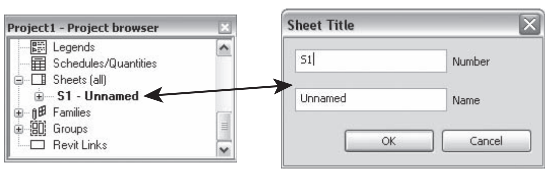


FIGURE 12.2
Right-clicking a sheet view and selecting Rename is a quick way to number and name sheets.

Sheets have their own category in the Project Browser, which helps keep the sheets organized and easy to work with while documenting the model. Chapter 2 discusses methods of creating browser view types to help sort views so you can easily maintain their organization. Sheets are separate from other views, but you can use these same methods for organizing your sheets. To start the creation of a new browser view type for sheets, go to Settings ➤ Browser Organization and select the Sheets tab shown in Figure 12.3. See Chapter 2 to learn how to further create new types and maneuver between them.

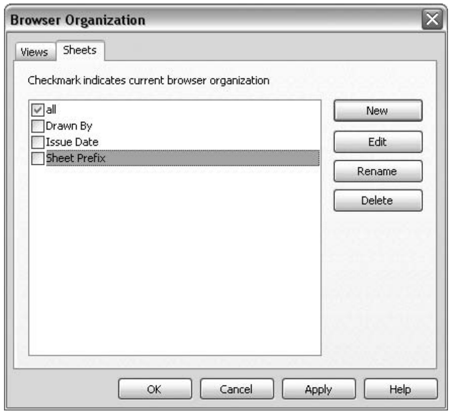


FIGURE 12.3
Create Project Browser view types for sheets to organize sheets in the Project Browser.

A sheet view, which is basically a drawing sheet, is where you will place your titleblock for displaying project-specific information with regard to location, owner, and design team as well as sheet numbering, sheet naming, and specific issue information. Like any other object or element in Revit Structure, a titleblock is a family. In this case it is an external family and can easily be created to match your current titleblock standards and/or to match another standard your client may require.

Creating a Titleblock

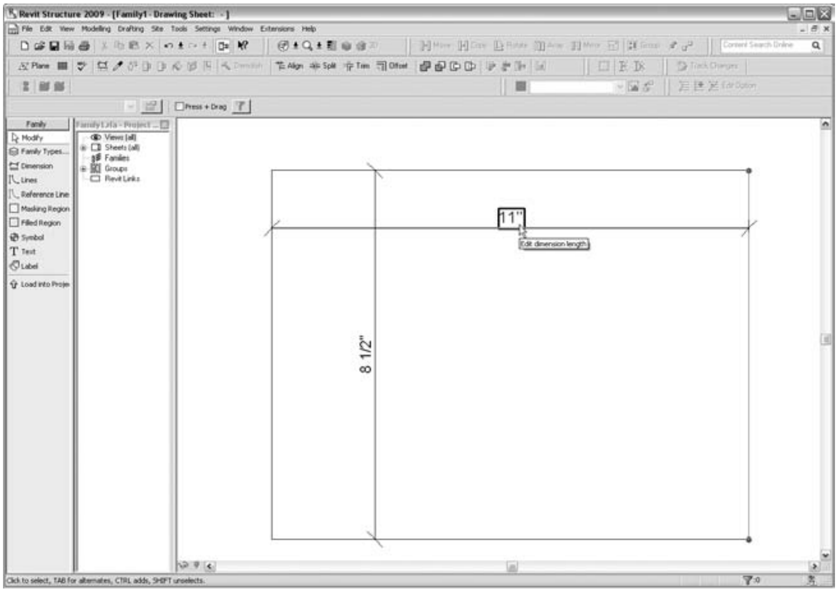
Revit Structure has sheets and titleblocks. They need each other in order to work properly, and to Revit Structure they pretty much mean the same thing. Since sheets are a system family, all information pertaining to them must be created within the project. Some information and its behavior are hardwired to the Drawing Sheets category. We will discuss some of these aspects later in this chapter. Creating the titleblock for a sheet allows you to display this system information on a sheet along with any other required data that pertains to the project. A titleblock is an external family, so you can add such information as line work, annotation, images for logos, and revision schedules.

The basic procedure for creating a titleblock is as follows:

1. Choose File ➤ New ➤ Titleblock.
2. The Titleblocks folder opens, showing a list of available titleblock family templates. If you do not see these files, browse to the Imperial Templates - Titleblocks folder and look for them there.
3. Select a size-defined template, or select the `New Size.rft` file and click Open.

4. You will see the extents of a titleblock, with dimensions indicating its size, as shown in Figure 12.4. You can adjust this line work to the extents of the titleblock. Once the extents are defined, you can delete the dimensions or turn them off to allow a clear drawing space.

Figure 12.4
Adjust the size of your titleblock by selecting the line work and changing the dimension values.



5. Choose Settings ➢ Family Category and Parameters, and you will see that the new titleblock family is part of the Title Blocks category and that it has only one hardwired parameter within the family.

6. Right-click within the drawing area and select View Properties, and you will see that the titleblock family already displays project parameters and shows how it behaves in a drawing list and revision schedule when it is placed into a project.

7. Save your new titleblock family, and proceed to adding line work, annotation, logos, and any other information that may be needed.

After you define the size for your titleblock, you can add content to it such as line work, text, labels that hold project information, and images such as company or project logos. This content can be imported and used from existing CAD drawings or created from scratch. Once your titleblocks are complete, they can be parametric to allow prompt changes.

Adding Line Work to Your Titleblock

Line work can be added by using the tools in the Design bar while in the Family Editor. You can use tools such as Lines, Masking Regions, Filled Regions, and Symbols to help place line work. Lines are defined by a subcategory of the Title Blocks category, which is located in the Annotation Objects tab of the Object Styles dialog, as shown in Figure 12.5. You can create a new line style by choosing Settings ➢ Object Styles.

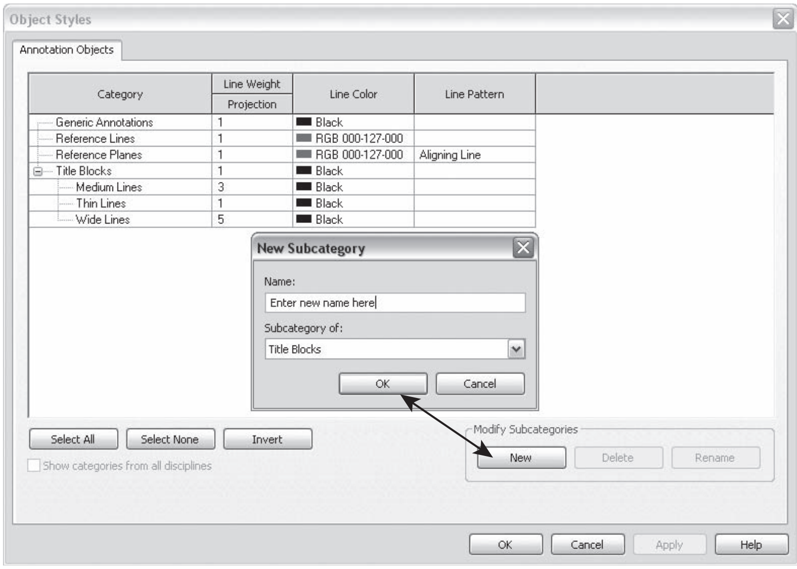


FIGURE 12.5 Create a new line style for titleblocks in the Object Styles dialog.

Make sure that the Subcategory Of value is set to Title Blocks, or the new style will not be available in the Type Selector. Existing 2D CAD data can be imported into the titleblock family to aid you in the layout of your line work. You can trace over the top of the imported line work, or you can explode the imported CAD data, which will convert the line work into Revit Structure lines. Creating native line work is relatively simple, so try to avoid leaving unexploded imported objects in the titleblock family. Convert everything to Revit Structure content so the behavior is a bit more predictable and the family remains empty of unnecessary information.

Some borders may have thicker lines to define separations within the titleblock or to define a boundary. In this case you can use a filled region set to a solid fill rather than using a wide line weight, as shown in Figure 12.6.

FIGURE 12.6 Filled regions can be used to show extra-thick lines in a titleblock family

MAKE YOUR ANNOTATION INTELLIGENT

You can easily add text to your titleblock by using the Text tools located in the Design bar in the Family Editor. You can create text styles and use them just as they are used in the project model environment. For each different type of text, you will need to create a new style. New text styles that are created in a titleblock family will not be available to the project when it is loaded in. This allows you to create the various styles that are needed for the titleblock and have the comfort of knowing that they will not be cluttering up your project model. Using the Text tools to place text does not allow them to display the properties of the drawing sheet or any project

information. To display this type of information you should use the Label tool. Figure 12.7 shows an example of text and labels being used together in a titleblock.

FIGURE 12.7 Use Text tools where text remains unchanged, and use Label tools where information is set and changed in the project.

Adding labels to your titleblock families allows text to automatically adapt to changes that are made to the information that they display. These labels can be used to display information such as Drawn By, Checked By, Sheet Name, Sheet Number, and other parameters. Most of these already exist as project parameters or are built into the titleblock family. If you need to add additional parameters, you can create them yourself. Chapter 2 discusses adding a project parameter for the contractor's name and having it display in the titleblock. The same procedure can be applied to other unique parameters you need to create.

To get the intended behavior when creating labels, make sure to take a moment to determine whether they should be an instance or a type parameter. For example, the Project Issue Date that is automatically created in the Project Information dialog is a type parameter, and the Sheet Issue Date within the titleblock family is an instance parameter. All project parameters will be type parameters, and the parameters that are already defined in the titleblock family are instance parameters.

Another method for making your annotation, as well as other forms of line work, intelligent is to create a Yes/No parameter that allows you to turn the display of those objects on and off. The basic procedure to achieve this is as follows:

1. While in the titleblock family select Family Types from the Design bar.
2. Click Add from the Family Types dialog to start the creation of a Yes/No instance parameter.
3. Fill in the appropriate information, as shown in Figure 12.8. Name it **Not for Construction**, make it a Yes/No parameter type, group it under Graphics, and make it an instance parameter.
4. Once the parameter is created, select the elements within the drawing area that you want to toggle on and off, and select Element Properties from the Options bar. In this case we will be selecting a piece of text that reads "Not for Construction."
5. In the Element Properties dialog select the small box to the far right of the visible parameter.
6. Select the Not for Construction parameter from the Associate Family Parameter dialog box, and click OK to return to the Element Properties dialog.

7. Observe the equal sign that now displays in the small box to the right of the parameter. This indicates that the element properties are linked to the Yes/No parameter. Figure 12.9 shows the process involved to connect the text to the Yes/No parameter.

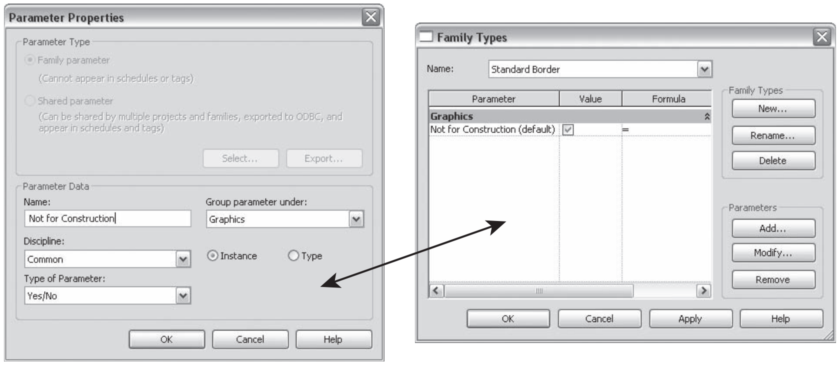


FIGURE 12.8 Add a Yes/No parameter to control the visibility of a Not for Construction note.

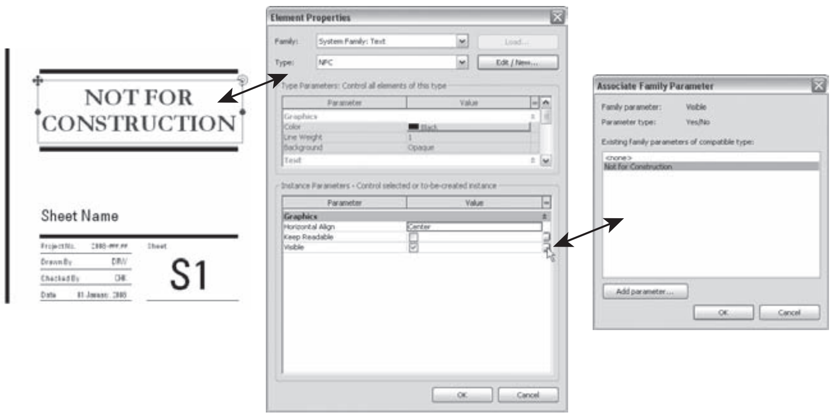


FIGURE 12.9 Create a Yes/No parameter to help control the display of elements in a titleblock family.

Depending on how you want the Yes/No parameter to behave, you can make it an instance or a type parameter. If you designate it a type parameter, you could have two different titleblock types: one For Construction and the other Not for Construction. If it were an instance parameter, each instance of the titleblock family could have its own unique setting.

Another good use for a Yes/No parameter in a titleblock family is in creating a line work grid, as shown in Figure 12.10. This grid can be set to display as an instance parameter of the titleblock family to be displayed and used while you are placing and arranging views on a sheet. After you have finished placing the views, you can toggle off the detail grid so it will not plot. Revit Structure will help align views to each other, but this grid can help prevent users from trying to place too many views on a sheet as well as aid in complying with sheet layout standards.

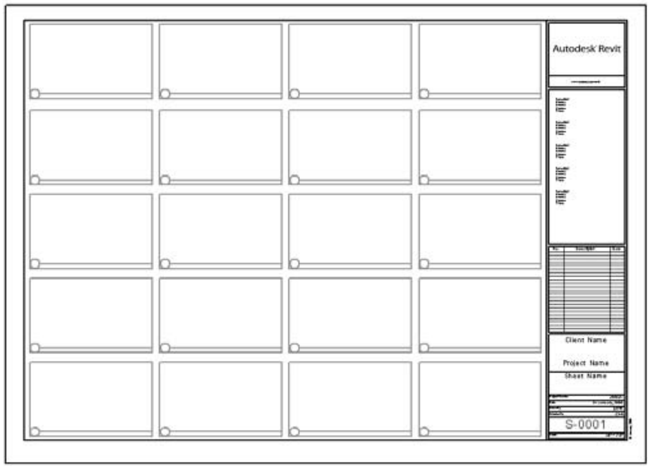


FIGURE 12.10
Make a grid of line work intelligent by placing it on a Yes/No Instance parameter.

FINISH IT OFF WITH AN IMAGE OR LOGO

Revit Structure allows you to import several different image formats into your titleblock families. You can see a list of these formats as well as the first step in importing an image by choosing File ➢ Import/Link ➢ Image. Figure 12.11 shows a logo image imported into a titleblock. One of the properties of an image is Lock Proportions; this ensures that the image maintains its aspect ratio when rescaling. This setting can be found in the image's properties or within the Options bar when the image is selected.

FIGURE 12.11
You can adjust the size of an image by selecting it and then dragging the grips at its corners.

You'll have better luck using images in Revit Structure for logos that contain fancy font display rather than using a complex filled region with line work and adjacent text, custom fonts, or imported 2D CAD graphics. If images are not created for these complex logos, you can perform screen captures with the Windows Ctrl+Alt+Print Screen keys and then paste the screen capture into editing software to crop as needed. This may not always produce a high-quality graphic, however, so you may need to use other image-capturing or printing tools.

INCORPORATE A REVISION SCHEDULE

Chapter 11 discusses in depth how to create various schedules for quantities and for displaying structural information for documentation. In this section we will discuss the things that are different in revision schedules from the schedules that are created within the project environment.

Revision schedules can be created only while in a titleblock family. Therefore this feature will be grayed out in a project environment. Revision schedules behave and are created pretty much the same way as other schedules created in the project environment. When revision schedules exist in a project, they populate with revision information as it is created and added to the project. Adding this revision information is discussed in the section "Keeping Track of Revisions."

The basic procedure for creating a revision schedule is as follows:

1. While in a titleblock family choose View ➢ New ➢ Revision Schedule.
2. By default, parameters that are available for schedule fields are already present, as shown in Figure 12.12. At this point you can click OK, and a schedule will be created.

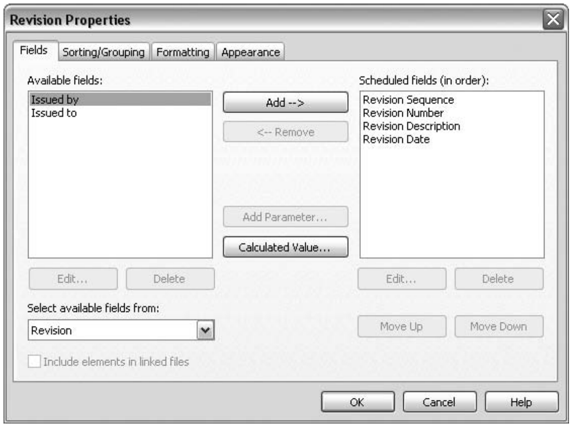


FIGURE 12.12 You cannot create new parameters for a revision schedule.

3. Observe the available tabs at the top of the Revision Properties dialog shown in Figure 12.12. These tabs all have the same look and feel that other schedules have. The only difference is that revision schedules do not have a Filter tab. Therefore, you will not be able to use filters when creating these types of schedules.
4. Once the schedule is created, it will appear in the Project Browser, as shown in Figure 12.13.
5. There are a couple of ways to get the revision schedule onto the titleblock sheet. One way is to select and drag the revision schedule in the Project Browser over to the hyphen that is located under Sheets (all). When you release the mouse button, the drawing area will switch to the view that contains the titleblock line work; there you can place the revision schedule just as you would any other schedule. Another way is to already have the titleblock open for editing and then drag the revision schedule from the Project Browser onto the sheet.

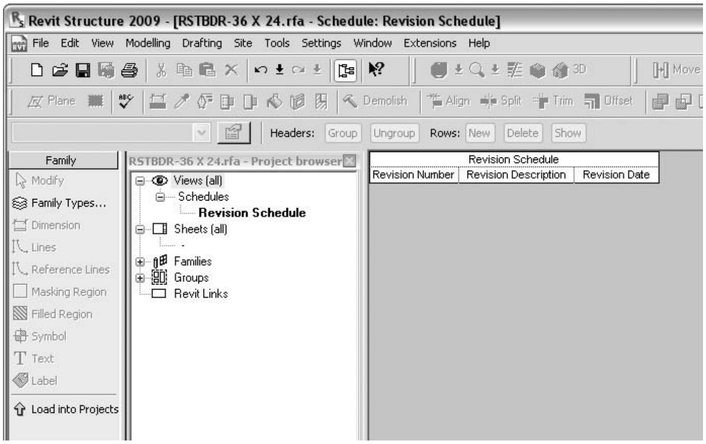


Figure 12.13
Revision schedules show up in the Project Browser, where you can drag them onto the titleblock sheet for use in your project

6. Adjust the appearance of your schedule in its Element Properties dialog, and adjust its size to fit within your titleblock by selecting the revision schedule while in your titleblock sheet view and dragging the shape handle control as required.

Getting a Revision Schedule to Populate

Only schedule revisions when they are placed into a project where cloud elements exist. Clouds have to become part of a sheet before the revision schedule can recognize them as a revision. To do this, you must place views containing clouds on the sheet with the revision schedule or place clouds directly onto the sheet.

Earlier we said that creating a revision schedule is pretty much the same as creating other schedules within a project. Let's discuss those aspects that are different by looking at the Appearance tab, shown in Figure 12.14. You can find this tab by going to the properties of the revision schedule.

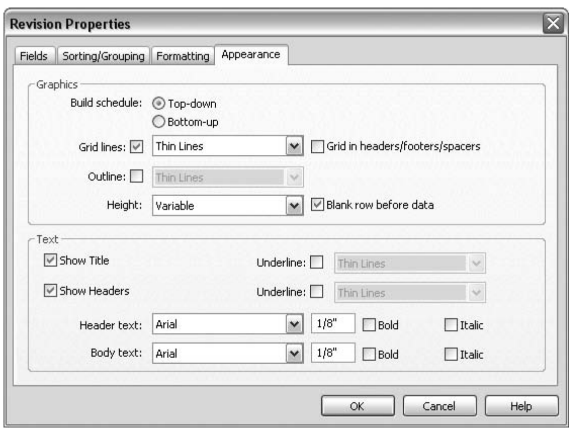


Figure 12.14
Revision schedules offer more functionality when it comes to their appearance.

Within the Appearance tab shown in Figure 12.14 you will find fields that are available only for revision schedules. These appearance settings allow you to build the schedule to autopopulate from the top down or from the bottom up or have a variable, user-defined height. This information will display for other schedule types but will be grayed out and unavailable.

Building Your Schedules Top Down or Bottom Up

The first field that will become available is the Build Schedule: Top-down Or Bottom-up. This allows you to define the schedule to display schedule information from the top down, like any other schedule that is created in Revit Structure. Selecting Bottom-up allows you to create revision schedules that list the scheduled information starting from the bottom. Figure 12.15 shows the end result of using these features.

FIGURE 12.15
Schedules can be set to be displayed Top-down or the Bottom-up.

No.	Date	Revision Description
1	05/20/2008	Revision 1
2	06/11/2008	Revision 2

2	06/11/2008	Revision 2
1	05/20/2008	Revision 1
No.	**Date**	**Revision Description**

Figure 12.15 also shows the schedules with a user-defined height that allows them to show only a fixed number of rows so they can be placed in a defined area inside the titleblock.

Building Your Schedules with Height Adjustment

The second field that will become available is the Height field, which allows you to define the schedule height as Variable or as User Defined. Setting the Height value to a variable height allows the schedule to display as many horizontal rows as required. The height of the schedule grows as each new revision is added to a sheet. With this setting your schedule has a potential to grow to an undesirable height or length that might eventually conflict with other annotation and/or line work in your titleblock.

When your titleblock requires a limited space to be available for revision information, you should set the Height value to User Defined. This allows you to adjust the overall height of the revision schedule while inside the titleblock family. Figure 12.16 shows a selected schedule with its Height set to User Defined.

FIGURE 12.16
When Height is set to User Defined, you can adjust the fixed height by dragging the grip control.

No.	Date	Revision Description

Schedules that have the Height value set to User Defined can display only a certain number of revisions depending on the number of rows available to display the revision information. As the schedule fills up, the earlier revisions get pushed out to make room for the new ones. If a schedule is created to only show five rows of revision information and it is filled with five revisions, revision 1 will be removed from the schedule to make room for revision 6. When revision 7 is added, revision 2 is removed.

If I Could Only Rotate My Revision Schedule on the Sheet

Well, you can. While in a titleblock family with the revision schedule already placed onto a titleblock sheet, select the revision schedule geometry and review the Options bar. You should see a drop-down list labeled Rotation On Sheet, which allows you to rotate the selected revision schedule 90 degrees clockwise or counterclockwise. This same feature is available for other schedules that are placed inside your project.

Revision schedules can be created to accommodate many different requirements. The best way to see all the various options is to create a new titleblock, add a revision schedule, and experiment with all of the different options. Load the titleblock into your project, and add revisions to the sheets to see how each change to the various settings in the Sorting/Grouping, Formatting, and Appearance tabs in the revision schedule properties behaves. Adding revisions to sheets will be discussed shortly.

Adding Information to Your Sheets

Sheets are one of the ways to disperse information about your project so others can access it. Typically these sheets will eventually be plotted. Beyond adding a titleblock to your sheets, you can add any view that was created that displays bits and pieces of the model in a schedule, section, or plan form. Placing these views onto sheets is really as simple as dragging them from the Project Browser and dropping them onto the sheet view. Once they are placed on a sheet, additional parameters become available as properties to the view. Some of these parameters are unique to the view type that is being placed, and others like the Sheet Number and Detail Number are part of almost all types of views.

Revit Structure's use of view titles helps keep your views organized onto sheets. View titles can display information about the view so those using the sheets can easily find the information they are looking for.

Sometimes thinking outside the box will allow you to take advantage of Revit Structure's views and pull information from the model. Creating a key plan from the model is an example. As the model changes, your key plan changes also. Adding more line work or text on the sheet as an overlay to the views that are placed on it allows you to provide additional information that is specific to the sheet or to combine two separate details into one. You can even overlap plan or elevation views to create nonexistent composite views.

Placing and Working with Views on Sheets

Different view types are used depending on what type of information you are showing about the model and how you want to show it. These view types are system families. Learning the parameters that appear when these views are placed onto a sheet and the different behavior each one can take on will help you produce great-looking documents that are easy to understand and work with.

There are several ways that you can add views onto a sheet. The various methods for adding any view onto a sheet are as follows:

Method 1 With the sheet view already open, drag and drop the view from the Project Browser onto the sheet.

Method 2 Drag and drop the view from the Project Browser onto the sheet name in the Project Browser. This automatically opens up the sheet view to allow you to place the view.

Method 3 Within the Project Browser, right-click the sheet onto which you will be placing a view, and select Add View. This action automatically opens to the selected sheet view and displays the Views dialog box shown in Figure 12.17. This dialog box shows a list of views that are not yet placed onto any sheet. Choosing a view and clicking the Add View To Sheet button allows you to place the selected view onto the sheet. By using this method you can avoid having to search through other views that are already placed onto sheets.

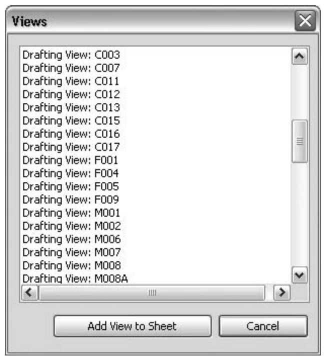


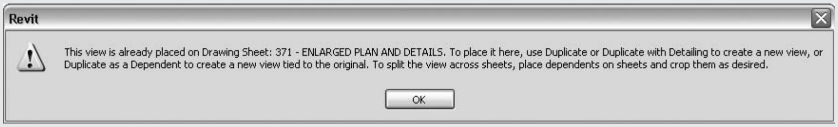
FIGURE 12.17 Selecting from a list of views that are not placed onto sheets

Method 4 With the sheet view already open, go to the View tab on the Design bar and select Add View. The same dialog box shown in Method 3 appears, where you can select from a list of views. The active view must be a sheet view, or the Add View To Sheet button will be grayed out.

As you can see, there are several methods you can use to place views onto a sheet. It is up to you to determine which one works best with your working habits. Once views are placed onto sheets, you can take advantage of those parameters that become available or those that have information available for display while they are on the sheets.

Only One Sheet Please

Revit Structure does a great job of keeping track of what views are on sheets, the sheet number they are on, and their detail number on the sheet. This helps keep your documents coordinated at all times. For this reason Revit Structure allows some views to be placed onto a sheet only once. If the same view was on a sheet twice, it would be tough to keep track of which sheet it needs to be referenced to. When a user attempts to place one of these views onto a sheet that is already on a sheet, the warning dialog shown here will appear.



Adding views to sheets adds another level of organization to your project. When you expand any of the sheet views in the Project Browser, every view that is placed on that particular sheet is listed, as shown in Figure 12.18. Once you expand the list, you can select any of the views and right-click it to access its properties, rename it, remove it from the sheet, or open it to edit.

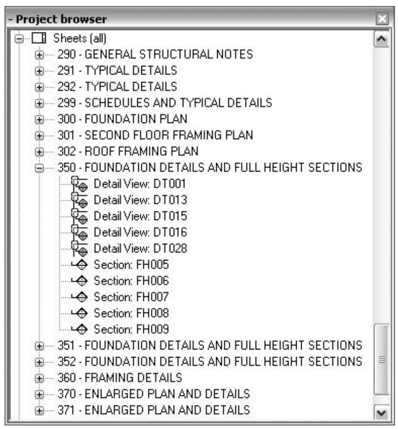
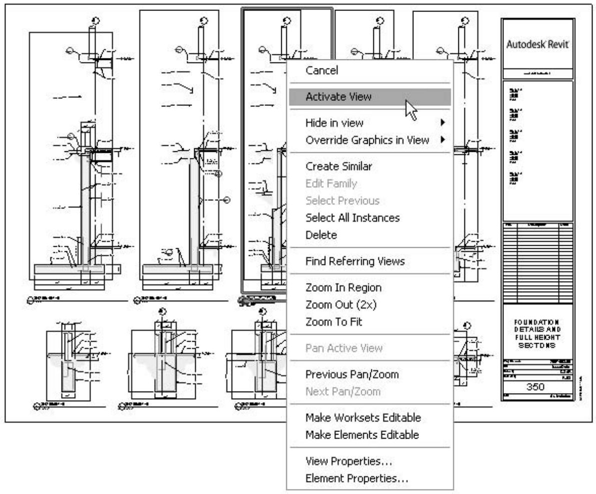


Figure 12.18
Use the sheet views in the Project Browser to quickly access views that are placed on sheets.

You also have the ability to access the individual views that are placed on a sheet by activating a view and working through the sheet. Once you are further along in your project, this can be an efficient way to access the views that you are using for documenting the model. Figure 12.19 shows that you can right-click on any view that is on a sheet to select Activate View from the context menu. This is no different from opening up the view from the Project Browser except you are seeing it displayed on your sheet with other views as well as working in the view.

When you have finished working in the view, right-click again anywhere on the sheet and select Deactivate View. This can be an excellent method for reviewing and editing your documents while avoiding plotting the sheets or if you are picking up minor red lines that were made on a hard copy or a design review markup.

Figure 12.19
You can work on views through your sheets by right-clicking a view and selecting Activate View.



Be Sure to Deactivate Your Views or Halftone Can Happen

If you are in the process of activating and deactivating your views to work in them, Revit Structure will automatically deactivate the previous view when a new one is activated. There is no need to deactivate a view before moving onto the next one. If a new one is not activated and you forget to deactivate the current view, Revit Structure will keep it activated on that sheet. If you plot a sheet and all views on that sheet but one appear to be halftoned, then you might want to check to see that all views are deactivated.

Understanding the behavior of the various views as they are placed on sheets and learning which parameters are associated with each are definitely something that you want to master. Grasping things like the various view properties, view title manipulation, and key plan strategies will give you the edge you need to use every bit of information that is available to you to help create a good-looking set of documents that will complement your modeling efforts.

Plans, Elevations, Detail, Drafting, and 3D Views

These view types have all the additional properties applied to them when they are placed onto a sheet. If for any reason some of them need to be rotated on a sheet, you can rotate them clockwise or counterclockwise by using the Options bar and the Rotate On Sheet drop-down list at the time of placement or after the view has been placed. The value of the Title On Sheet parameter can be filled in to override the view name that is displayed in the view title.

Other parameters that become available for these view types are as follows:

Detail Number This contains the value that the view is given as its number on the sheet. The user can change this value, but no number can exist twice on the same sheet. When the first view is placed, +17 Structure gives it a value of 1. As additional views are placed on the sheet, Revit Structure automatically takes the next number in line or the lowest number not used.

Sheet Number This contains the value of the sheet number that the view is placed on. This value updates as a view is moved onto a different sheet or a sheet number changes.

Sheet Name This contains the value of the sheet name that the view is placed on. This value updates as a view is moved onto a different sheet or a sheet name changes.

Referencing Sheet This contains the value of the sheet number that the view is referenced from. If a view is referenced from more than one sheet, then the sheet number that referenced the view first will be displayed as the value.

Referencing Detail This contains the value of the detail number that the view is referenced from. If a view is referenced from more than one detail number, then the view that referenced the view first will be displayed as the value.

The parameters mentioned here can be used to display information in the view titles to aid in the layout and control of referencing information while you are documenting the model. Legends, schedules, and Graphical Column Schedules have behaviors that are much different from those of other views when they are placed on sheets.

Legends

Legends are somewhat a combination of a detail view and a drafting view as to what is displayed inside of them, but they resemble the properties of schedules and Graphical Column Schedules when they are placed on sheets. They are like drafting views to the extent that they do not allow you to display geometric information directly from the model and they do not have a view range, but like other views they can have annotations, symbols, region fills, detail lines, and the like added to them. They are similar to detail views in that they do allow you to display model information, except that information is not graphically from the model. You can display parametric information about the various family types that exist throughout a project. A more in-depth explanation on legends and keynote legends can be found in Chapters 9 and 11.

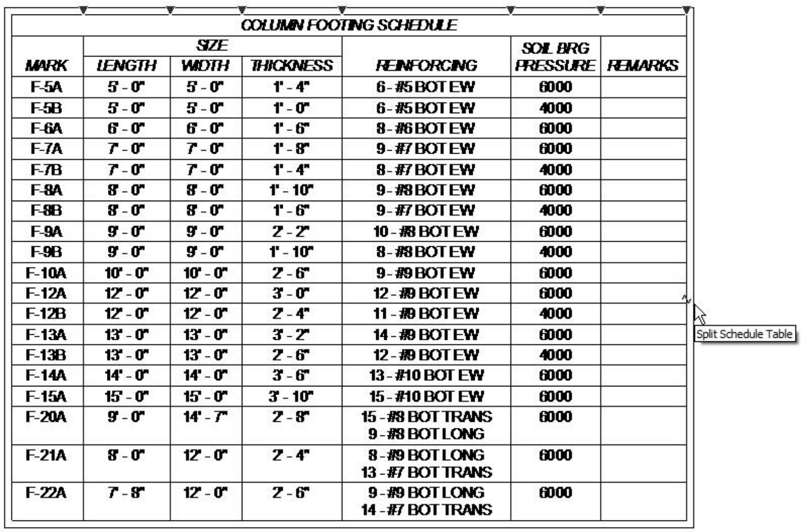
Legends cannot be rotated when they are placed on a sheet, and they are not referenced to sheets. Since they are not referenced to sheets, they do not have the additional parameter to keep track of what sheet number they are on or what number they are assigned when placed on a sheet. They can be placed onto multiple sheets but cannot be placed multiple times on the same sheet. Revit Structure does not support multiple instances of a legend on the same sheet. You can override the display of a legend's view name in a view title by filling in the value for the Title On Sheet parameter.

Schedules

Chapter 11 discusses how to create and work with schedules prior to them getting placed onto a sheet. Once a schedule is placed onto a sheet, it can be rotated clockwise or counterclockwise 90 degrees by selecting the schedule and using the Rotate On Sheet drop-down list on the Options bar. A schedule can be placed onto the same sheet multiple times; therefore it will not have a Detail Name parameter assigned to it when it is placed, nor will it have any other parameters.

Selecting the schedule while it is on a sheet and choosing Element Properties will only bring up a blank properties box.

Once a schedule is placed on a sheet, you can adjust the width of the columns by dragging the shape handle controls that display when the schedule is selected. You can do this only after the schedule is placed on a sheet. If for any reason a larger schedule needs to be split into separate chunks to fit on a sheet, you can easily do so by selecting the schedule while it is on the sheet and clicking the break symbol at the right of the schedule. Each click will break the schedule in equal parts. Figure 12.20 shows a single schedule that is selected and about to be split into two segments.



COLUMN FOOTING SCHEDULE						
	SIZE				SOIL BRG	
MARK	LENGTH	WIDTH	THICKNESS	REINFORCING	PRESSURE	REMARKS
F-5A	5' - 0"	5' - 0"	1' - 4"	6 - #5 BOT EW	6000	
F-5B	5' - 0"	5' - 0"	1' - 0"	6 - #5 BOT EW	4000	
F-6A	6' - 0"	6' - 0"	1' - 6"	8 - #6 BOT EW	6000	
F-7A	7' - 0"	7' - 0"	1' - 8"	9 - #7 BOT EW	6000	
F-7B	7' - 0"	7' - 0"	1' - 4"	8 - #7 BOT EW	4000	
F-8A	8' - 0"	8' - 0"	1' - 10"	9 - #8 BOT EW	6000	
F-8B	8' - 0"	8' - 0"	1' - 6"	9 - #7 BOT EW	4000	
F-9A	9' - 0"	9' - 0"	2' - 2"	10 - #8 BOT EW	6000	
F-9B	9' - 0"	9' - 0"	1' - 10"	8 - #8 BOT EW	4000	
F-10A	10' - 0"	10' - 0"	2' - 6"	9 - #9 BOT EW	6000	
F-12A	12' - 0"	12' - 0"	3' - 0"	12 - #9 BOT EW	6000	
F-12B	12' - 0"	12' - 0"	2' - 4"	11 - #9 BOT EW	4000	
F-13A	13' - 0"	13' - 0"	3' - 2"	14 - #9 BOT EW	6000	
F-13B	13' - 0"	13' - 0"	2' - 6"	12 - #9 BOT EW	4000	
F-14A	14' - 0"	14' - 0"	3' - 6"	13 - #10 BOT EW	6000	
F-15A	15' - 0"	15' - 0"	3' - 10"	15 - #10 BOT EW	6000	
F-20A	9' - 0"	14' - 7"	2' - 8"	15 - #8 BOT TRANS 9 - #8 BOT LONG	6000	
F-21A	8' - 0"	12' - 0"	2' - 4"	8 - #9 BOT LONG 13 - #7 BOT TRANS	6000	
F-22A	7' - 8"	12' - 0"	2' - 6"	9 - #9 BOT LONG 14 - #7 BOT TRANS	6000	

FIGURE 12.20
Split a schedule into two by clicking the break symbol at the right of the schedule.

Figure 12.21 shows the same schedule split into two segments. Dragging the grip control and moving it up or down will add or subtract rows from one segment to the other. Dragging the move control back onto the adjacent segment will join the schedules back together. Clicking the break symbol again will split the schedule into additional segments.

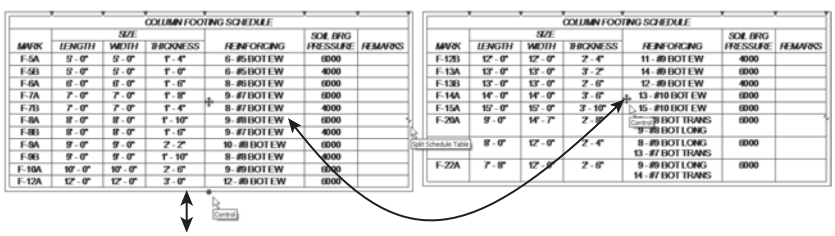


FIGURE 12.21
Split a schedule into two by clicking the break symbol at the right of the schedule.

Graphical Column Schedules are also schedules, but are assembled and automated much differently by Revit Structure. They too can be split into multiple segments but take a much different approach than just clicking a break symbol.

Graphical Column Schedules

Chapter 4 discusses in great depth how to create the Graphical Column Schedule (GCS) as well as several of its properties. The GCS has a behavior that it takes on only when it is placed onto a sheet. The only way to access this information is to select the GCS that is on the sheet and choose Element Properties. Accessing the properties by first selecting the view name in the Project Browser will not display this information. Figure 12.22 shows the additional segment information that is displayed under the Extents group of the element properties.

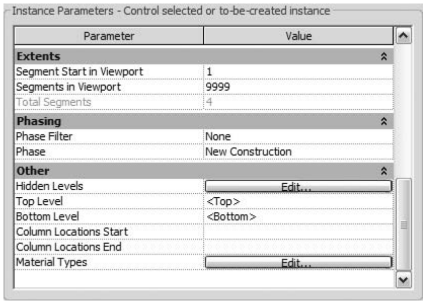


Figure 12.22
Place segments of a Graphical Column Schedule onto multiple sheets by adjusting the segment parameters in its properties.

The GCS does not have a Detail Number parameter when it is placed onto a sheet, so it can be placed onto different sheets multiple times but not twice onto the same sheet. Revit Structure does not support multiple instances of a GCS on the same sheet. In the properties shown in Figure 12.22, the GCS has a total of four segments. The Segments Start In Viewport and Segments In Viewport parameters are instance parameters for each GCS that is placed on a sheet. This means each parameter can have its own value and not affect the other one but still exist as one GCS. In some cases all of the segments may not fit onto one sheet, so these values will have to be adjusted accordingly.

With the help of Chapter 4, create a five-segment GCS. The basic procedure for distributing a GCS segments onto multiple sheets is as follows:

1. Create two sheets. The first one will contain three segments, and the second one will contain two segments of a five-segment GCS.
2. Place the five-segment GCS onto each sheet.
3. Make the first sheet view active, and choose the GCS properties by selecting the GCS on the sheet and then accessing its properties.
4. In the Extents group of its properties change the value of Segments In Viewport to **3**. This will display segments 1 through 3 for this instance of the GCS. Click OK to see the results.
5. Make the second sheet view active, and choose the GCS properties by selecting the GCS on the sheet and then accessing its element properties.
6. In the Extents group of its properties change the value of Segments Start In Viewport to **4** and the value of Segments In Viewport to **5**. This will display segments 4 through 5 for this instance of the GCS. Click OK to see the results.

Note that if you set the initial extents of the viewport values while the GCS is placed on only one sheet, Revit Structure will automatically display the remainder of the segments when the next instance of the GCS is placed onto another sheet.

As you can see, these parameters allow you to adjust each instance of the GCS so you can display whatever segments you want to display as the GCS is placed onto a sheet. Normally the view title displays the view name when a GCS is first placed on a sheet. This view name can be overridden by filling in a value for the Title On Sheet parameter. If a value exists for this parameter, the view name that is displayed in the view title by default is replaced with it. The GCS has a parameter to display its own title at the top of the schedule, so the view title will more than likely be set to not display.

Working with View Titles

When views are placed on sheets, view titles are used to display information about those views. These view titles are linked directly to the view and pull information directly from the properties of the view. As properties of the view change, the information that is displayed in the view title automatically changes with it.

Views titles can have different types assigned to them to allow you to display different information about a view throughout a sheet. Some views may require only a title, and some may not require anything. Each required display is a new view title type. Three view title types exist in the `Structural Analysis-Default.rte` template that is provided by the Revit Structure installation.

- No Title
- Title Only
- Title w Line

These view titles are set to display the scale, the view name or title on sheet, and the detail number inside a circle. You can create additional view title types by duplicating an existing one and changing its properties as needed. If you want to display new geometry and/or parameters, then you will have to create a new view title family. For instance, some companies prefer to add the sheet number as part of their view titles.

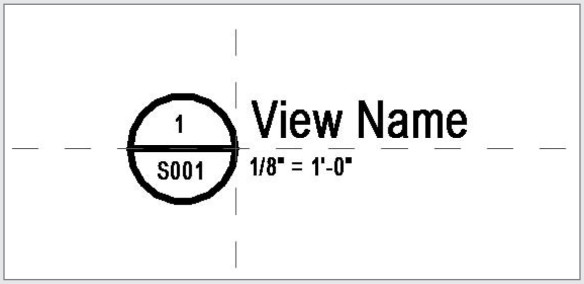
Exercise: Creating a New View Title

To create a new view title you can start from an existing family that is already part of the Revit Structure installation, or you can start from scratch by using a View Title template. Revit Structure already has a family created similar to the one we are creating, except the circle element is larger. You can locate this family by browsing to the Imperial Library - Annotations folder.

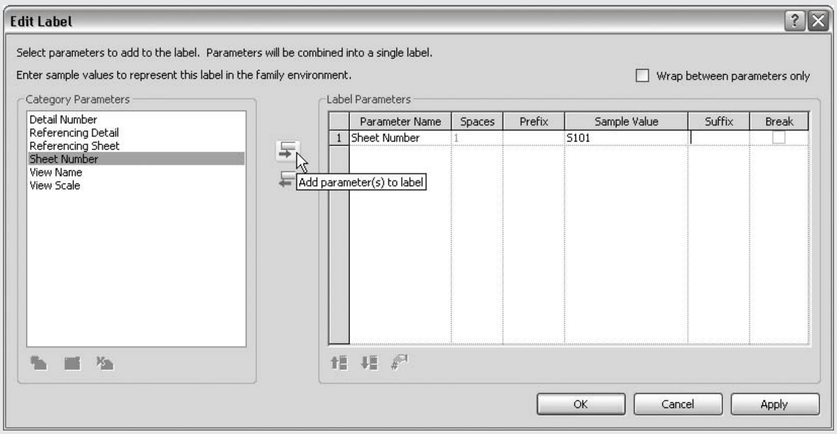
For this exercise we will start from a View Title template so you can see the steps involved for creating one from scratch. To start we will create the look of the view title in a family and then load it into the project, where we will make it a View Title type.

Creating a New View Title Family

1. Choose File ➢ New ➢ Family.
2. Browse to the `View Title.rte` family template located in the Imperial Templates - Annotations folder.
3. Choose Settings ➢ Families Category And Parameters. Observe that the template already has the category set to View Titles.
4. Add line work and labels as shown in example graphic by using the tools on the Family tab in the Design Bar.



5. For the circle and line geometry, use the Lines tool in the Family tab along with various types of lines, circles, or arcs that can be found in the Options bar after choosing Lines.
6. To place the parameter information for the text that will be part of the view title, click the Label tool located in the Family tab. Once you select Label, click anywhere in the drawing area to display the Edit Label dialog box, where you can choose any available parameter.



7. To add a parameter, choose it in the Category Parameters area at the left, and click the Add button. This is the button with the arrow pointing to the right. Selecting this button places the parameter in the Label Parameters area to the right. Click OK to apply the parameter and close the dialog.

8. Once the label is placed in the drawing area, drag it into its final position. Check to make sure its justification properties are set properly.
9. Repeat steps 5 through 8 to add the other labels to the view title.
10. Delete the red block of note text that was preloaded in the template, and save the family, naming it whatever you wish. After saving, load it into your project.

Creating a New View Title Type

1. In your project select a view title or view that is displayed on a sheet, and go to its properties.
2. Click the Edit/New button, and choose Duplicate.
3. In the new View Title field type a new name and click OK.
4. Change the Title parameter value to match the view title family name that was created previously for the view title family by selecting the family name from the Title drop-down list.
5. Make adjustments, if needed, to other graphic parameters such as Line Weight, Color, Line Pattern, and the visibility of its Title or Extension Line.
6. Click OK in the Type Properties box, and click OK to close the Element Properties dialog.
7. Observe the new view title type, and check to see that its appearance is as you intended.

View titles allow you to control the display of the various components that compose them in a couple of places. To control the line weight, line color, and line pattern of the extension line, you will need to go to the type properties of the view title, as shown in Figure 12.23.

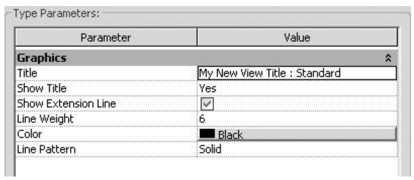


Figure 12.23 Change the display of a view title extension line by going to the properties of the view type.

To control the line weight of the annotation symbol that is created inside the family, you will need to choose Settings ➤ Object Styles, as shown in Figure 12.24.

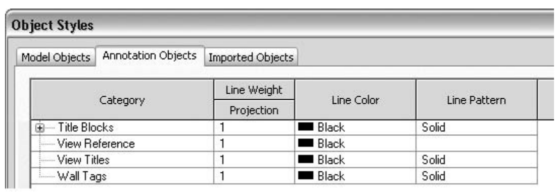


Figure 12.24 Change the appearance of view title elements by using Object Styles.

Within the Annotation Objects tab of the Object Styles dialog, go to the category View Titles. Here you can change the line weight, line color, and line pattern.

Key Plan Strategies

In the past in the 2D CAD environment, key plans probably consisted of line work that made up the perimeter shape of a building, some annotation for area delineation, and hatching to denote which area is shown on the sheet. There were probably other reasons for key plans even though you were not referencing zones or areas of a large-footprint building. You can still use those same strategies in Revit Structure, or you can go one step further and use modeled geometry to produce your key plans. Using modeled geometry for key plans will help keep them up to date with the current design of the building. It may even help you use key plans more often to aid in producing clearer documentation of your model.

There are several strategies for creating key plans, and they are explained in the following sections. Each strategy has pros and cons. You will need to choose which strategies will work for you as an office standard or on a project-by-project basis.

Reduced Structural Plans

Using a reduced structural plan is probably the best strategy to allow the key plan to update as the model updates. The idea is that separate plan views are created at each level and then placed on sheets. The following procedure lists the steps to take to quickly create key plans by using structural plans.

1. Create a structural plan of a level by creating a new one or duplicating an existing one. Name it accordingly.
2. Change the scale to a custom scale that allows a small key plan to be placed in a convenient and consistent location on a sheet.
3. In the Visibility/Graphics Overrides dialog of the view, turn off the display of all model and annotation categories by selecting all and toggling their visibility off. Only a few categories will be turned back on. With everything off it is much easier to turn on only the visibility of those model categories that you want to be displayed.
4. Turn on the visibility of only those categories that you want to be displayed. Usually Walls, Floors, and Roofs will be enough categories to give you the display of the building footprint. Categories can be toggled on or off depending on how you want the key plan to display.
5. Override the line weight of the categories that are on to accommodate the reduced scale.
6. If needed, hide individual elements in the view or override their display individually.
7. Once everything is set the way you want, create a view template and name it **Key Plan**.
8. Create additional key plans for other levels, and apply the view template Key Plan to them.
9. Add additional line work, annotation, and region fills as required.
10. Drag each one to its appropriate sheet.

When using this method, you will need to create separate key plans for each zone or area of the plan you will be denoting. Structural plans can be placed on only one sheet before they have to be duplicated. The example shown in Figure 12.25 is a key plan for Zone 2 First Floor Framing Plan. We created this by following the procedure just outlined. Each zone is its own separate key plan with annotation, line work, and region fills specific to each view.

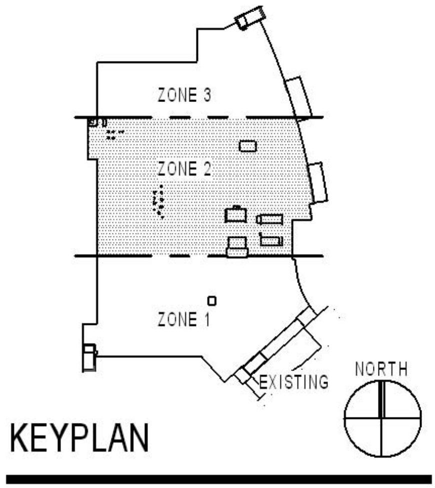


FIGURE 12.25 A key plan using a structural plan view to reference the Zone 2 framing plan

Figure 12.26 shows a similar example of a key plan displaying the parametric section cuts to reference the location of core wall elevations that were used to call off construction joint locations, wall reinforcing, and embed plate locations. Everything automatically updates as the model changes or sections change locations.

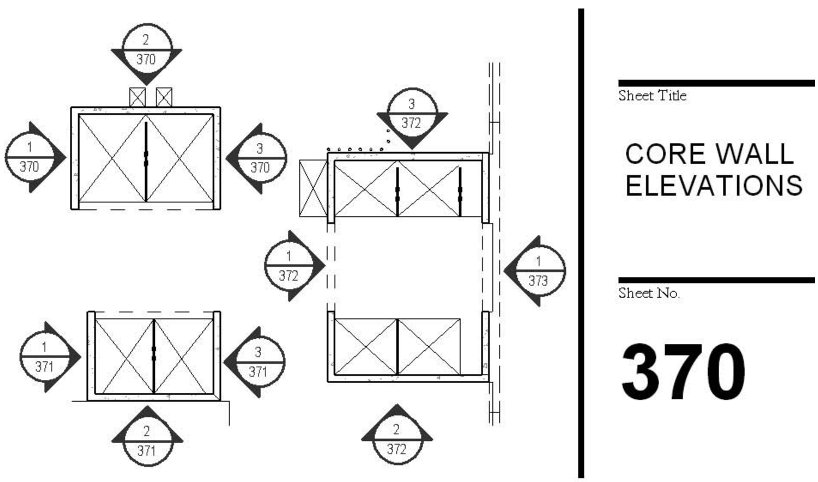


FIGURE 12.26 A key plan using a structural plan view to reference core wall elevations

Legends

Legends can also be used to create key plans. When using legends you are restricted to using only the drafting tools that are available in the Drafting tab of the Design bar. If a key plan is already created in another CAD system, the line work can be imported into the legend to be used directly or to be traced over the top of. Since legends can be placed onto more than one sheet, there is potential to reduce the number of key plans you have to create. If it is not critical to follow the exact footprint, the same Zone 1, Zone 2, and Zone 3 legend could be placed on levels 1 through 10. In this scenario only three legends would have to be created. Figure 12.27 illustrates the use of a legend as a key to show what the information that is being displayed in a Graphical Column Schedule relates to. Using a legend as a key in this instance allows you to have one key that can be placed on multiple sheets.

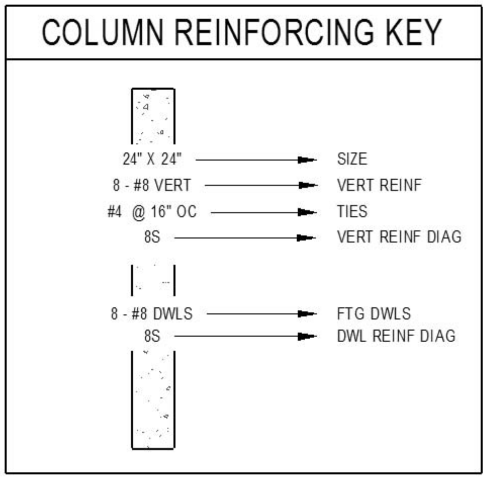


Figure 12.27
Using a legend for a key diagram allows the same view to be placed on multiple sheets.

Annotation Symbols

Making use of annotation symbols can be a clever little way to go about creating a key plan. This allows you to create a key plan family that can be placed as a symbol onto each sheet. Line work can be added just as you would in any other example shown here. If a key plan is required to show different shades to denote zones or areas, you can add a Yes/No parameter to control the display of the hatching or filled region. An example of how to create a Yes/No parameter can be found in "Making Your Annotation Intelligent."

The basic procedure for starting a new annotation symbol family is as follows:

1. Choose File ➢ New ➢ Annotation Symbol.
2. Browse to the `Imperial Template\Annotations` folder, and select the `Generic Template.rft` template.
3. A symbol used as a key plan can be categorized as Generic Annotations. You can verify this setting by choosing Settings ➢ Family Category And Parameters. Highlight the Generic Annotations category, and click OK.

If the example shown in Figure 12.25 for the reduced plan method was created by using an annotation symbol, you could create the line work by placing new line work or importing existing line work. You could create a reduced plan view by using elements from the 3D model. The reduced plan view could be exported to AutoCAD to create 2D line work. This AutoCAD file could then be imported into the annotation symbol family. For the separate zones, region fills could be created and placed on Yes/No parameters. Three parameters would be created for this (Zone 1, Zone 2, and Zone 3). Each one would be set accordingly to create its own type, so depending on where the key plan was placed, it could display Key Plan Symbol Type Zone 1, Zone 2, or Zone 3.

Image or Screen Captures

Images and screen captures can be used for key plans as well. Screen captures or images can be developed from existing key plans and placed into other views that were previously mentioned. Additional annotation, line work, and region fills can be used in conjunction with these images to create a look that suits your needs.

Creating a Sheet Index

Revit Structure automatically keeps track of sheets, and information such as sheet name and number can be displayed in a schedule-like format called a drawing list, more commonly known as a sheet index. As shown in Figure 12.28, creating a sheet index is similar to creating any other schedule in Revit Structure in that you can add fields, create filters, provide sorting or grouping, adjust the formatting, and set the appearance.

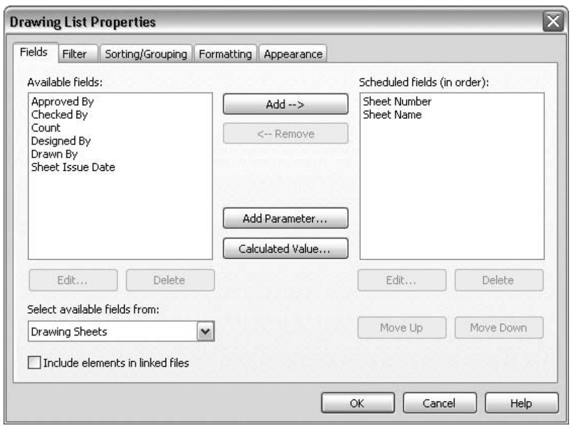


FIGURE 12.28 Creating a drawing list is similar to creating schedules or quantities.

You can create and manipulate drawing lists using the same methods as for the schedules discussed in Chapter 11, except that when you create a drawing list you use a specific Drawing List tool that locks the category to display only information that is part of the Drawing Sheet category.

The basic procedure for creating a sheet index is as follows:

1. Choose View ➢ New ➢ Drawing List.
2. Within the Fields tab of the drawing list properties, add any of the available fields on the left-hand side of the dialog to the scheduled fields on the right-hand side by selecting the field and clicking Add.
3. To include sheets from another Revit Structure file that is linked to the project, check Include Elements In Linked Files.
4. Reorder the scheduled fields if needed by selecting a field and clicking the Move Up or Move Down button as required.
5. Create any filters and sorting or grouping rules that may be needed.
6. Adjust the formatting of the sheet index headings and the alignment of the scheduled text.
7. Set the appearance of the graphics and text.
8. Click OK to finish the drawing list.

Drawing lists can also be useful in helping to keep track of titleblock information. Viewing or populating this type of information can be easily be done through a schedule form rather than by individually viewing sheets and their properties. Information that can be added to a traditional sheet index to be used in checking for titleblock coordination includes the following:

- Checked by
- Designed by
- Drawn by
- Approved by
- Issued date

Sorry, You Can't Be in My Drawing List

Sheets can selectively be chosen to not display in a drawing list. To do this, go to the properties of the sheet that is not to be displayed, and remove the checkmark from the Appears In Drawing List parameter field located in the Identity Data group. Checking the box allows that particular sheet to be displayed in a drawing list, while deselecting it prevents it from being displayed.

Sometimes you may need to create a much more advanced sheet index for larger projects or for those that have issue after issue. Keeping track of these issues can otherwise be a nightmare. Using Revit Structure to help automate some of this work will help eliminate confusion as to which sheets were issued when and for what reason. Figure 12.29 shows a more advanced drawing list that keeps track of the sheets as well as what they were issued for.

Figure 12.29
A sheet index created with additional parameters to keep track of issued sheets

STRUCTURAL SHEET INDEX							
		ISSUE					
SHEET #	SHEET TITLE	50% CD REVIEW	95% CD REVIEW	ISSUED FOR CD	ADDENDUM NO. 1	ADDENDUM NO. 3	AS-BUILTS
290	GENERAL STRUCTURAL NOTES	X	X	X			X
291	TYPICAL DETAILS	X	X	X			X
292	TYPICAL DETAILS	X	X	X			X
299	SCHEDULES AND TYPICAL DETAILS		X	X	X		X
300	FOUNDATION PLAN	X	X	X	X		X
301	SECOND FLOOR FRAMING PLAN	X	X	X			X
302	ROOF FRAMING PLAN	X	X	X	X		X
350	FOUNDATION DETAILS AND FULL HEIGHT SECTIONS		X	X	X		X
351	FOUNDATION DETAILS AND FULL HEIGHT SECTIONS		X	X	X		X
352	FOUNDATION DETAILS AND FULL HEIGHT SECTIONS		X	X			X
360	FRAMING DETAILS		X	X			X
370	ENLARGED PLAN AND DETAILS		X	X		X	X
371	ENLARGED PLAN AND DETAILS		X	X		X	X

You can create a more advanced sheet index by adding additional project parameters that are linked to the Drawing Sheets category. Once these parameters are added they will become available to be added to the scheduled fields. Within the Formatting tab, the headers for the Issue fields are set to display as vertical. When sheets are issued, the value of the Issue parameter in the properties of the sheet is populated with an *X* or any other character or symbol that you choose. Once the sheet index is placed on a sheet, you can adjust the columns as required.

The basic procedure for creating these additional parameters is as follows:

1. Go to the properties of the drawing list, and in the Fields tab, click Add Parameter.
2. Select Project Parameter as the parameter type.
3. Give it a name of **ISSUE_01_50% CD REVIEW**.
4. Select Text for the type of parameter.
5. Select a group to place the parameter in that makes sense to you.
6. Click OK.
7. Repeat steps 1 through 6 for as many parameters as you want to add.

Parameters

Additional parameters can be added ahead of time and displayed in the schedule until used or hidden within the schedule until they are used. To hide a parameter, go to the Formatting tab within the drawing list properties, select the parameter to hide, and check the box named Hidden Field.

8. Once the new parameters are created, they are available to be added as scheduled fields.

You can create a sheet index that also keeps track of the issue date information with the addition of a drafting view or a legend. This method requires that the drafting view or legend be manually coordinated with the drawing list. Figure 12.30 shows a sheet index that has been created with this method.

FIGURE 12.30
A sheet index created with additional parameters and a drafting view to help support the issue information

STRUCTURAL SHEET INDEX

SHEET #	SHEET TITLE	ISSUE														
		A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
290	GENERAL STRUCTURAL NOTES	•		•	•	•	•									
291	TYPICAL DETAILS	•		•	•	•	•									
292	TYPICAL DETAILS	•				•	•		•							
300	FOUNDATION PLAN - ZONE 1	•		•	•	•	•	•		•		•				
300.1	FOUNDATION PLAN - ZONE 2												•		•	
301	PENTHOUSE FLOOR FRAMING PLAN - ZONE 1	•	•			•	•		•	•		▲				•
301.1	ROOF FRAMING PLAN @ INFILL - ZONE 2										•				•	
302	PENTHOUSE ROOF FRAMING PLAN	•	•			•	•		•	•		▲				
303	PARTIAL PLAN AND DETAILS								•							
304	PENTHOUSE CONNECTOR													•		
340	FULL HEIGHT SECTIONS						•			•		•				•
360	SECTIONS AND DETAILS	•		•	•	•	•	•	•							
361	SECTIONS AND DETAILS			•	•	•	•	•		▲						•
362	SECTION AND DETAILS					•	•		•							•
363	SECTIONS AND DETAILS							•								
364	SECTIONS AND DETAILS									•		•				
365	SECTIONS AND DETAILS @ INFILL										•				•	
366	SECTIONS AND DETAILS @ INFILL												•		•	
380	COLUMN SCHEDULE AND DETAILS	•	•			•	•		•							
381	COLUMN SCHEDULE @ INFILL AND MISC DETAILS														•	

KEY FOR ISSUE

ISSUE	ISSUED FOR	ISSUE DATE
A	PRICING PACKAGE	NOVEMBER 15, 2007
B	REVISED PRICING PACKAGE	DECEMBER 03, 2007
C	BID PACKAGE #3 - FOUNDATION AND UNDERGROUND	DECEMBER 10, 2007
D	PERMIT REVIEW SET	DECEMBER 12, 2007
E	BID PACKAGE #4 - SHELL	DECEMBER 13, 2007
F	ADDENDUM #1	DECEMBER 20, 2007
G	ASI #1	JANUARY 21, 2008
H	SI #2	FEBRUARY 22, 2008
I	SI #3	MARCH 20, 2008
J	STEEL PACKAGE (INFILL)	APRIL 23, 2008
K	SI #9	APRIL 28, 2008
L	BID PACKAGE #8 (INFILL FDN)	MAY 02, 2008
M	SI #11	MAY 06, 2008
N	BID PACKAGE #9 (INFILL)	MAY 09, 2008
O	SI #15	MAY 16, 2008

• INDICATES DRAWINGS THAT ARE BEING ISSUED.

▲ INDICATES DRAWINGS THAT HAVE BEEN ISSUED ON SMALL SHEETS OR WRITE UP.

The upper portion of the sheet index shown in Figure 12.30 is from a drawing list, with additional parameters added to the Drawing Sheets category. Symbol characters are added for values of these parameters depending on how the sheets are being issued. One way to do this is to insert a symbol via the character map and copy and paste it into the text field within Revit Structure.

The lower portion of the sheet index is a drafting view, with line work and text added to match the look of the sheet index. This portion has to be manually coordinated with what is shown in the sheet index above it each time you issue it. This method works well because you can display a much lengthier issue name without the sheet index getting too large, and you can also keep track of the issue dates as well as how the sheets were issued.

Learning to create a sheet index to this extent will help you to keep your sheets in sync with the revisions that you are adding to them. It may be necessary to manually keep track of these issues and revision names in a key schedule as already shown for your sheet index, but you will be able to use Revit Structure's revision tools to help automate the process on a sheet-by-sheet basis as well as automate the control of the information that gets displayed in a revision schedule.

Keeping Track of Revisions

Keeping track of revisions so others can clearly see what has changed with the model or with the documentation is another task that Revit Structure does well. Similarly to how views on sheets are repeatedly being coordinated with the sheet they are on and where they are referenced from, revisions know what sheet they are placed on, keep track of whether they are issued or not, as well as hold other valuable information about the changes being made.

A typical revision consists of a cloud that holds all the information that pertains to a particular revision. The cloud is placed around the area of change to indicate where the change is made. A tag that displays information from the cloud properties is placed alongside the cloud. When Revit Structure detects that a revision is on a sheet and a revision schedule is present in the titleblock, it displays and tracks the revision information for that sheet. Figure 12.31 shows revisions as they would be used on a project.

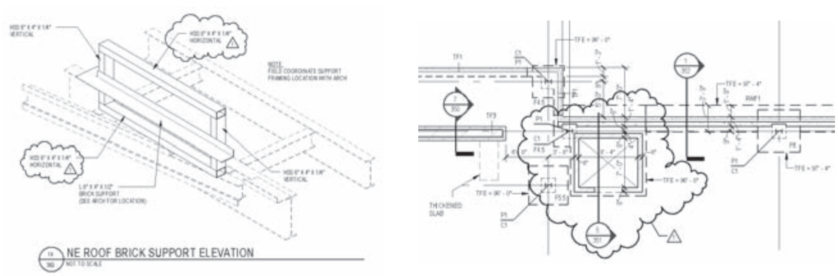


FIGURE 12.31 Keeping track of revisions by placing a cloud and tagging it for reference

The basic procedure for placing and tagging a revision is described in the following sections.

Placing a Revision Cloud

Placing clouds in Revit Structure is very similar to how you may have done it in a prior 2D/3D CAD environment. You may have used an inserted block or a tool that allowed you to sketch the extents of the cloud. Revit Structure's Revision Cloud tool allows you to sketch the boundary of the revision cloud directly in the views where the revisions occur or directly on the sheet. To place a revision cloud, follow these steps:

1. Choose Drafting ➢ Revision Cloud, or on the Design bar click the Drafting tab and select Revision Cloud. You can also right-click an existing revision cloud and choose Create Similar from the context menu.

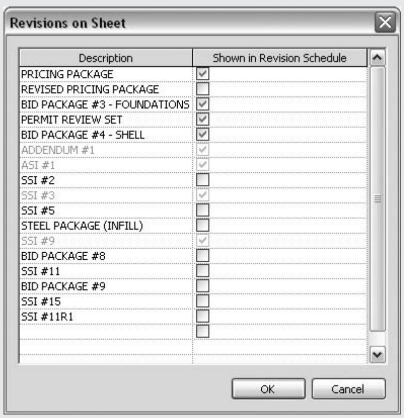
2. Placing a cloud will put you into sketch mode, where you sketch the extents of the cloud by using the Line tool.
3. Clouds are sketched by consecutively clicking placement points to form a boundary. Each sketch point clicked creates a double set of arcs to represent a cloud shape. Pressing the spacebar while sketching an arc set reverses the direction of the arcs.
4. When the sketch is complete, click Finish from the Sketch tab.

One of the best features of placing revision clouds in Revit Structure is that revision clouds contain properties that allow you to assign them to defined revisions. These properties allow Revit Structure to keep track of their location on a particular sheet. You can tag revision clouds just like other elements in Revit Structure to display their properties or denote them as a symbol that links to a revision schedule, where additional information can be displayed.

No Revision Cloud Required

Revit Structure automatically populates a revision schedule with revision cloud information when it detects a revision being placed on a sheet. Sometimes you may want to record information about a revision within a titleblock that does not require a revision cloud. For example, you may want to record information about a 50 percent review set, issued for DD, or issued for CD. Whatever it may be, it might not require a revision cloud.

You can create a revision on a sheet without requiring a cloud by right-clicking a sheet in the Project Browser and choosing Properties. In the properties of the sheet click the Revisions On Sheet parameter located under the Identity Data group, and click the Edit button. The following illustration shows the Revisions On Sheet dialog, where you can choose to record a revision on a sheet without having to place a revision cloud.



All revisions that are in the project will display, and those that are already on sheets from the placement of a revision cloud will be grayed out. Checking the Shown In Revision Schedule field populates the schedule with that particular revision without your having to place a revision cloud.

If multiple sheets are selected, this parameter will not be available to edit. You can select and set only one sheet at a time to display revision information.

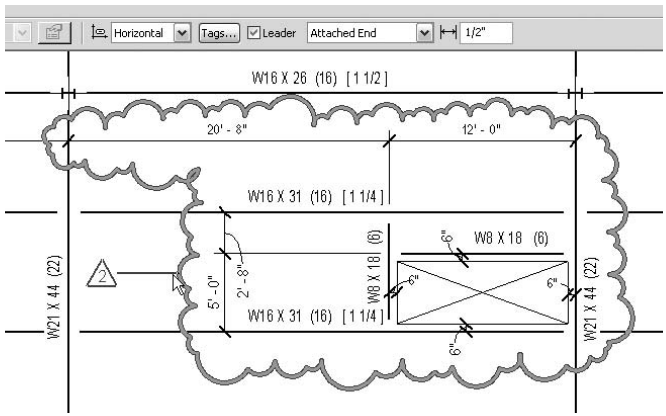
Tagging a Revision Cloud

Before you can place a tag to reference a revision, you will need to place a revision cloud. The tag is a label that can be defined by the user to display properties about the revision cloud. To tag a revision cloud, follow these steps:

1. Choose Drafting ➤ Tag ➤ By Category, or on the Design bar click the Drafting tab and select Tag ➤ By Category.
2. Use the Options bar to adjust placement properties prior to placement.
3. Select a revision cloud element to be tagged.

The Options bar shown in Figure 12.32 offers several options that you can set prior to placing a tag to a revision cloud. A similar Options bar can be found by selecting an existing tag.

FIGURE 12.32
Use the Options bar to toggle tag leaders on or off before and after their placement.



Reading from left to right, the Revision Tagging Options bar in Figure 12.32 allows you to do the following:

- Select Horizontal or Vertical to predetermine the orientation of the tag.
- Click the Tags button to select which loaded tag to use. If the desired tag is not loaded, you can load it from the Tags dialog.
- Choose to place a tag with or without a leader.
- Choose to have the end of the leader attached to the cloud by setting it to Attached End or to a location of your choice by setting it to Free End.
- Set the initial leader length dimension from the end of the leader back to the tag. After the tag is placed, you can move it along with the leader to any location you want.

Setting the Display of Revision Clouds and Tags

You use Revit Structure's object styles to set the display of revision clouds and tags that may be used to label them. These settings can be found by choosing Settings ➤ Object Styles. Within the Annotation Objects tab of the Object Styles dialog, go to the category Revision Cloud Tags And Revision Clouds. Here you can change the line weight, line color, and line pattern for each.

Making the settings in the Object Styles dialog is a global change, but you can apply the override for each view, giving you added flexibility in how revision cloud elements display.

You will eventually need to move revision clouds that are already placed throughout the model to a different revision. If revision clouds are tagged, they will automatically be updated with the new revision information. You can do this by selecting the revision cloud(s) and choosing Element Properties or by going to the Options bar. Figure 12.33 shows that either way allows you to choose which revision type the cloud should be assigned to.

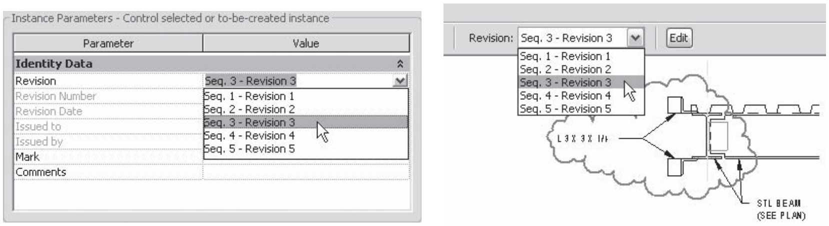


FIGURE 12.33
Change revision cloud types by going to its properties or the Options bar when it is selected.

If you need to move several revision clouds to a different revision, you may need to individually select each revision cloud or select them all by using a window selection and filter the selection to select only the revision clouds. The Select All Instances method is not available when working with revision clouds in Revit Structure.

To understand the properties of revision clouds and how their properties are controlled, you will need to learn how to work with the Sheet Issues/Revisions dialog that Revit Structure uses to help set up various behaviors that may be required for a project's environment.

Understanding the Sheet Issues/Revisions Dialog

You access the Sheet Issues/Revisions dialog by choosing Settings ➢ Revisions. Within this dialog, shown in Figure 12.34, you can add additional revisions, merge with existing ones, as well as adjust the order in which they occur. You can mark revisions as being issued and set the display to show only the tag or both the cloud and the tag. Within the tag you can choose to display a numeric or an alphabetic value. All of this gives flexibility to you when tracking revisions. We will briefly discuss the behavior of these global settings within a project.

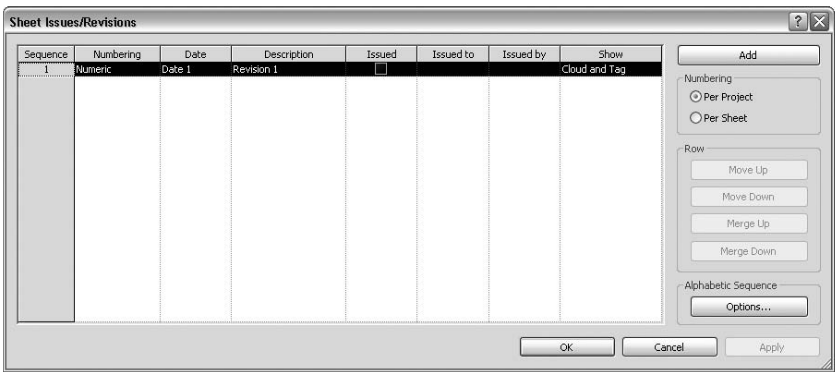


FIGURE 12.34
Creating new revisions, organizing their location, and controlling their behavior can be done in the Sheet Issues/Revisions dialog.

What? No Tag Loaded

Tagging a revision cloud is the same as tagging other objects in Revit Structure. The traditional tag for a revision is noted by a triangular shape, but you can revise it just as you would other tags to another shape or to display a range of information from the revision cloud's properties. Attempting to place a tag for a revision cloud without loading a matching tag will display a warning that indicates that a tag is not loaded for this type. You can load the default tag that comes with Revit Structure by browsing to the `Imperial Library\Annotations\Revision Tag.rfa` family.

When opening the Sheet Issues/Revisions dialog for the first time, you will see that Revit Structure has already created a revision for you. You can use this as a starting point. Each revision has a fixed number of parameters available. New parameters cannot be added, so sometimes you may have to use an unused parameter field to be able to use a revision standard that your company employs.

For example, if you were to use AD1 in your revision delta symbol to indicate Addendum 1, you could use the Issued To or Issued By parameter to hold the value of AD1. You would have to create a new revision symbol to display the value from one of those parameters. Since these parameters are global to the project, you can change the value in one place and concurrently change the display of several instances of tags.

To help you better understand the various settings in the dialog shown in Figure 12.34, we will briefly explain what each section does. To further expand your knowledge from what is shown here, browse through the help guides that are part of Revit Structure by searching for "Revisions."

Sequence Revit Structure gives each revision a sequence number. This parameter can be added to a revision schedule to help sort the order of revisions placed on a sheet.

Numbering The Numbering parameter is used to select the display of numeric, alphabetic, or no numbering for each revision. This value is typically displayed in a revision delta. If a numeric number method is selected, it will display numbers in sequential order, such as 1, 2, 3, 4, 5, etc., as each revision is created.

If an alphabetic numbering method is selected, it will pull its information from the alphabetic sequence list shown in Figure 12.35. This list can be found by clicking the Options button in the lower right-hand corner of the Sheet Issues/Revisions dialog. Each revision will display the next character in line, such as *A*, *B*, *C*, *D*, *E*, etc., as each revision is created.

Note that the alphabetic sequence cannot contain spaces, numbers, or repeated characters. If you attempt to use this format, Revit Structure will display a warning to you saying such.

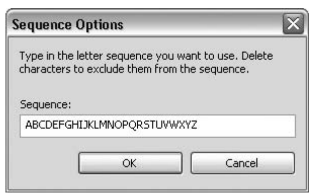


Figure 12.35
Use Sequence Options dialog to set the order for an alphabetic numbering method.

Date The Date parameter is used to display the date of issue for the revision in a revision schedule on a titleblock.

Description The Description parameter is used to display the revision description for the revision in a revision schedule on a titleblock.

Issued The Issued parameter is used to mark a revision issued. This locks the revision clouds that are placed throughout the model so that they cannot be moved, deleted, or edited. Issued revisions cannot be modified. This helps ensure that revision clouds don't get moved after they are issued. Keep in mind that the revision clouds do not automatically move when other modeled or annotation elements move, so it is possible to have the revision cloud not encompass its intended revision.

You cannot place a revision cloud on a revision level that has been issued. If you need to do such a thing, you will first need to mark the revision as not issued. When you place new revision clouds, they will automatically get placed on the last unissued revision in the revision list.

Issued To The Issued To parameter can be used to display information about whom the revision is being issued to. This field can sometimes be used as an available field to display information other than an Issued To value.

Issued By The Issued By parameter can be used to display information about who is issuing the revision. This field can sometimes be used as an available field to display information other than an Issued By value.

Show The Show parameter allows you to display the tag, the cloud and tag, or neither before or after it is issued. This usually offers enough flexibility so you can display the information that you need at the time you need to display it.

Maybe I Don't Want a Leader

Sometimes there are cases where you need to turn off the clouds after a revision is issued but keep the tag turned on during future revisions. This is commonly done so you can track revisions without having a cluttered mess of clouds on the sheets. If tags are placed with leaders and the Show parameter for that revision is set to show only tags, the leaders will also be visible. This can lead to an unexpected display of tags and leaders. You might want to rethink using leaders for your revision tags.

Numbering The Numbering area of the Sheet Issues/Revisions dialog box is where you can set the revision behavior to Per Sheet or Per Project. This setting can be set to however your company tracks revisions on sheets.

When using the Per Project method, each revision gets assigned a numeric or alphabetic value depending on what revision it is. No matter what number of revision may be on a particular sheet, it will always display the same value. For example, if revision 3 was the fifth revision in the Sheet Issues/Revision dialog, it would display the value 5. Wherever that revision was detected on a sheet or set to display on a sheet through the sheet properties, it would display the value 5.

When using the Per Sheet method, each revision gets assigned a numeric or alphabetic value depending on what revision it is on that particular sheet. Each revision is given a value depending on how many revisions are on a sheet. For example, if revision 3 was the fifth

revision in the Sheet Issues/Revisions dialog and was the second revision on a sheet, it would display the value 2.

Row The Row area of the Sheet Issues/Revisions dialog box is where you can rearrange the order of revisions or merge them with other ones. How often does it happen that you are working on revision 3 and you get the call, "Can we quickly issue this one detail as Revision 3 and put everything else you have been working on for the past three days on Revision 4?" Well, this is how you would handle it. You can add a new revision, move it up ahead of Revision 3, swap the names, and can continue on your way. In the past this may have not been such an easy task.

Alphabetic Sequence The Alphabetic Sequence options are used when the Numbering method is set to Alphabetic. This dialog sets the sequence to be used for alphabetic characters as revisions are placed on sheets.

Real World Scenario

Everybody Has a Standard

As a structural engineering firm, we work with several clients. Each one has its own standard for keeping track of revisions. Not only does each client have its own standard, but each project, depending on the construction schedule and contractor, may warrant a deviation from that standard. Revit Structure accounts for many of these standards, but in some cases we have to do a little bit of thinking outside of the box in order to use Revit Structure's Revision Schedules and Tracking feature.

In one scenario we create a revision in Revit Structure for a Structure Supplemental Issue (SSI). This can be a small item or a large item. They are usually issued as full-size sheets and tend to happen on a regular basis. The numbering system inside the revision tag takes the form SSI#2, depending on which SSI number it is. To get the revision tag to display this numbering method we have created a new revision tag to pull from the Issued To or Issued By parameter, which is a global parameter located in the Revision dialog. The field that is chosen is populated with the SSI# for the revision. This parameter is also added to the revision schedule so that the correct numbering is displayed in the schedule on the titleblock.

Sometimes these SSIs may take a few days to be completed, and during this time other revisions may have to be answered in another form. For us it is usually in the form of a Request for Information (RFI). As these RFIs are answered, and if the model needs to be revised, it is denoted with a revision cloud and tag. In this case we have created a new revision tag to pull from the instance comment parameter of the revision cloud. The comment parameter is populated with a value of RFI#10, depending on which RFI number it is. The revision cloud is still on an SSI revision, but the instance of the cloud would have an RFI comment. We use the revision comment tag for this.

When the SSI is officially issued, revisions not yet issued are tagged as SSI#, and those that have already been issued on smaller sheets are tagged with RFI#, or whatever method it was officially issued. This allows for a full-size sheet to display all the newly issued revised information as well as how it was issued and display the information that was previously issued but on partial sheets.

Typically Revit Structure should greatly reduce the need for RFIs because of the close collaboration we can have with other disciplines and the early detection of conflicts. However, with some projects we tend to quickly develop the model and may not always have all the discipline's correct information at the same stage of the race where we are structurally or have a Revit Structure/BIM model available to us. Keeping track of these revisions in this manner allows us to stay on top of what is or has gone on with our model as we continually keep the model up to date with the construction schedule.

You will find that learning how the tools in Revit Structure work for creating sheet content and tracking revisions will help you learn some workaround solutions that can be done to achieve the behavior of several standards that may be required. You will want to take full advantage of everything pertaining to your deliverables and what sheets can do for you so you can spend your time designing and documenting the structure rather than coordinating the placement of your information, as it may change as a project evolves.

The Bottom Line

Create a Titleblock to Display Project Information The basics of creating a titleblock include using line work, annotations, filled regions, labels, and images. Combining these basic elements to create parametric behavior will take you way beyond 2D drafting.

Master It What are three ways you can make your titleblocks parametric to autoadapt to changes that are made within your sheets?

Create a Revision Schedule to Your Company Standards Revision schedules added to titleblocks allow you to keep track of revisions on sheets. You can design these revision schedules to accommodate just about any company standard and titleblock configuration.

Master It How do you rotate a revision schedule 90 degrees? Where do you go to set a revision schedule to display its information from the bottom up?

Explore the Behavior of the Various View Types When Placed on a Sheet When views are placed on sheets, new parameters become available that display information that is specific to how and where the views are placed. Each view can have a different behavior; knowing this behavior allows you to take advantage of it.

Master It What are four parameters that become available in plan, elevation, detail, drafting, and 3D views when they are placed on a sheet? What types of views can be placed on a sheet more than once without being duplicated?

Produce a Sheet Index to Keep Track of Your Issued Sheets Revit Structure lets you easily create a sheet index to keep track of sheets.

Master It How can you add a second element to a sheet index to manually account for issue names and sheets that are being issued?

Control the Behavior of Revisions in Your Project You can control the tracking of revisions made to the model to reflect several standards that may be required as well as react to unknown project schedule changes.

Master It How do you set a revision tag to display an alphabetic numbering standard, and how do you rearrange a revision to be put on hold and placed behind another?

Part 4

Sharing Your Structural Model

Chapter 13

Worksharing

Prior to the introduction of Revit Structure, your structural projects more than likely consisted of several files that you assembled into sheets and printed for each issue. These files were probably produced and maintained by a drafter. Another set of files were created by the engineer for the analysis portion, which eventually was put into the documentation portion of the drawings. The use of all these files kept it fairly easy to keep the work spread out so multiple users did not need to have the same file open simultaneously. Even if one person was producing all aspects of the project, at some point a second user was often brought into the project at the tail end to help.

Now Revit Structure has taken all of your individual files that you used to work on and combined them into one database that consists of a single `.rvt` file. This database is now where all of your plans, details, and sheets exist. This file contains all your project documentation. Chances are, more than one person will have to work in this database. This is where *worksharing* comes in.

Worksharing allows your project to be broken up into logical pieces called *worksets*, which allow multiple users to work on the same project at the same time.

In this chapter you will learn to:

- Determine when to enable worksharing
- Enable and set up the worksharing environment
- Request and grant permission of elements
- Stay in sync with other team members
- Properly maintain your project file

Understanding the Worksharing Concept

With worksharing enabled, Revit Structure allows several team members to efficiently work on a project at the same time. Revit Structure does a good job of keeping track of ownership—in other words, who has which elements borrowed or checked out. This enables you to coordinate and propagate changes between users. Worksharing involves using worksets to create a multiuser working environment.

To truly grasp the concept of worksharing, you need to understand workflow, the various terms and concepts associated with worksharing, and when it makes sense to use worksharing and when it doesn't.

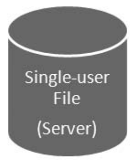
Alternative Method

For very large projects, models may be broken up into several smaller projects. Team members then work on individual smaller projects and link them back into another master project to produce the final documentation. Using this method will improve performance by not having everything stored inside one large working model—but at the cost of more project setup and management. This method can be used in conjunction with a multiuser worksharing environment or with just several single-user files.

Knowing the Workflow

A common scenario is where you, a single user, starts a project and takes it to a certain point just before having to enable worksharing. You set up your project and start the modeling process. Levels and grids are being coordinated with the architect, and generic object placeholders for walls, columns, framing, and so forth are added to the model. This single-user project is saved on your network server, as shown in Figure 13.1, so that it gets backed up, no different than how you save your individual files in the past.

Figure 13.1
A single-user file saved to the network



At some point during the project's schedule, additional users are needed. You may need someone to help with the analytical model or to help out with creating details. Every company and every project will have its own scenarios. Either way, you see that it is time for multiple users, and thus it is time to enable the worksharing feature.

Once you enable this feature, Revit Structure will take all of the content that you have placed into your project and assign it to logical worksets. The first stages of this organization are all handled by Revit Structure.

At this point you are no longer working on a single-user file. The only thing left to do is save your project so it becomes a central file for other team members to use. Save the central file onto the network just as you saved your single-user file.

Once the central file is created, users should create and save local files to their workstation's hard drive, as shown in Figure 13.2. These local files maintain a bidirectional link back to the central file on the network. Since each user opens and works off from his or her own local disk copy, multiple users can now work on the same project simultaneously.

As users create their own local copy and continue to work, they save their changes both locally and to the central file. Throughout this process, users stay in sync with one and other as they continue to work and make changes to the project.

This is only one possible scenario that can exist. You will need to determine for yourself the best workflow that works for you. Regardless of your workflow, the remaining chapter topics will help you get the process started.

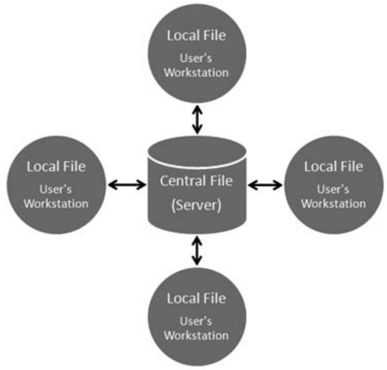


FIGURE 13.2
This worksharing diagram shows a central file saved to the server and local files saved to each user's workstation.

Speaking the Language

Before you get too much further, let's review some terminology. Don't worry: these terms will become second nature to you as you become more familiar with working on a project with worksharing enabled. You might even come up with your own definitions.

Worksharing Worksharing is the concept of using a combination of element borrowing and worksets to share the model.

Workset A workset is a grouping of elements that can be reserved for editing by a single user. These worksets usually consist of specific zones of responsibility, such as distinct wings or grouping of levels within building. Worksets can also be "checked out" so no other user can edit them.

Element borrowing Element borrowing is defined as an event that allows a person to edit an element without checking out worksets. When a user edits an element within the model through his or her local copy, only that element gets checked out/borrowed (not the workset it's on) from the central file by that user; that way, no one else can edit it until it is relinquished.

Checked out This is the effect of a user taking full ownership of a workset, which means no other user has rights to edit elements that are part of it. All elements that are part of a checked-out workset are now owned by the user who has them checked out.

Central file This is the type of `.rvt` file that Revit Structure creates once you enable worksharing. It becomes the master project file and contains a building model subdivided into functional areas through worksets. This master file is saved on the network so that more than one person can have access to work on the project.

Local file Local files are the `.rvt` file(s) used to enable worksharing. A local file is a copy of the central file that resides on a user's workstation's hard drive. This file maintains a bidirectional link back to the central file, which allows all users to interact with one another. Users work on the project by working on their local file.

Editing requests When one user tries to edit an element that is already being modified by another user, the user can place an editing request asking for ownership to modify the object.

Relinquish The best way to understand this term is to look up a synonym for it. If you were to do this, you would finds phrases and words such as *give up*, *surrender*, *hand over*, and

abandon. This is what you will do when you have finished working with borrowed elements from the project. You hand them back over, or *relinquish* them, so someone else can use them.

Save to Central (STC) This term will be used a lot in the worksharing environment. After you have been working on your local copy, which is linked to the central file, you will execute an STC to update the central file with your changes so other users can see them.

Reload Latest Just executing an STC does not automatically put your changes into other users' local copies. Reload Latest is a method to transfer changes that have been made in the central file back into your local file. Changes that may have been made in your local file do not get transferred to the central file by this method.

When to Use Worksharing

For the most part, almost every model should have worksharing enabled. More than likely you will have at least two people working within the model at some point in time, such as the engineer and the drafter/modeler. This number may vary depending on the size of the project.

Making the decision to work in a worksharing environment and set up worksets is greatly determined by the project size and the potential for multiple users working in the model at the same time. Even if multiple users are not in the project simultaneously, there are other benefits to be gained by enabling worksharing.

What's Up with This Dialog Box?

By default, Revit Structure assumes that you are working in a single-user environment. If you are working in your model and another user opens that model, that user will be presented with a dialog box indicating their copy will be in read-only mode. Seeing this dialog box on your computer screen frequently can become annoying, which might be a sign that worksharing should be enabled.

Looking at Project and/or Team Size

The project size can help determine whether worksharing should be enabled and how your worksets should be created. Prior to starting your project, discuss your options with regard to the project type and the number of potential simultaneous team members. These up-front discussions will allow you to take advantage of worksharing accordingly and allow proper setup for the following project types:

Small projects (two users) Worksharing should be enabled, but you should not have to enable additional worksets to divide up the model. As long as you are not routinely working on the same section or same area of the project, Revit Structure's automatic element borrowing should work fine for two users working on the project. An occasional editing request may have to be made.

Large projects (three-plus users) Worksharing should be enabled and logical worksets created. These worksets should reflect obvious breakpoints of a building (i.e., at expansion joints, discrete wings, grouping of levels). If additional users are brought on only to help create sections, then automatic element borrowing should be fine.

Large projects with multiple buildings and discrete wings Worksharing should be enabled as well as logical worksets created. For these types of projects, you may want to combine linked structural models and worksharing to help maintain more manageable files.

Suggested structural worksets are:

- By discipline (structural, architectural, MEP)
- By group of floors or groups of structural elements
- Project construction phase division
- Floor plan division at expansion/seismic joints
- By material type: steel, masonry, concrete, and so forth

These are only suggestions and certainly may not be specific in any way to how worksets should be created or teams set up for you. Knowing the simple logic behind determining worksets and potential roadblocks will help you put a basic standard in place.

Not Just for Multiple Users

During the beginning stages of a project when only a single user is required on a project, it *may not* be necessary to enable worksharing. However, there are several other reasons why you should think about enabling worksharing regardless of the type and size of your project. In these scenarios, additional worksets more than likely would not have to be created:

- Saving locally to your workstation's hard drive will tend to be much faster than saving across your network. Local saves can happen much more frequently.
- For inexperienced users, it may be necessary to allow more advanced users in the project to help perform advanced tasks or help maintain company standards within the model.
- It can be nice to allow inexperienced users to only work on sections and details rather than performing advanced modeling procedures. This serves as an excellent mentoring training method. However, understanding and using worksets is not a novice issue. It will require specific instruction and guidance.

There are several other reasons why you should enable worksharing regardless of being able to be in a multiuser environment:

- It allows you to have another level of graphic/element visibility display for elements that are not being worked on.
- It allows you to only load portions of your project that are needed for the task at hand, thus increasing computer performance.
- It allows you to have additional backups that are on both your local machine and your network.

If you are enabling worksharing to take advantage of the additional visibility settings and improve performance, you may need to create additional worksets. Keep in mind that if you create too many worksets for this purpose, your model can become unmanageable. Individual view overrides and the use of filters may be a better alternative.

Oh by the way, we had to separate our model

We had a project that had already gone out a few times for issue. Over a hundred different types of views made up our construction document set. The majority of these views were from the live model. For structural, the job was almost done and the architect had not yet completed a final issue.

The next time we received an updated model from the architect they give us an "oh by the way," we had to split the project into two files (Interiors and Shell) to increase performance. What did this mean to us? Well, whenever you link in a Revit model, it is turned on in every view by default. This meant that we had to go through every view and turn off the Visibility settings of the linked model.

Our solution was to create a workset, *Arch_Shell* and *Arch_Interior*. This allowed us to uncheck the Visible in Every View check box. It also allowed us to do selective opens of the linked models. (We will get into selective opening later in this chapter.) This technique was much better than unloading the link in the file. As you start collaborating with other disciplines, you might find that this little trick will work for you as well as open up many more possibilities.

Keep in mind that other disciplines that may be using your linked model will not be able to see elements from your model that are assigned to worksets with the "Visible in Every View" box unchecked.

Enabling Worksharing

Now that you have concepts and terminology down, let's get into enabling worksharing. There are a couple of ways to enable worksharing. One way is to select File ➢ Worksets.

Another way is to click the Worksets icon located in the Worksets toolbar (Figure 13.3). You can find this toolbar in the toolbar area at the top of the screen. Notice the grayed-out commands just below the Worksets command in the File menu and to the right of the icon in the toolbar. These commands are only available when worksharing is enabled.

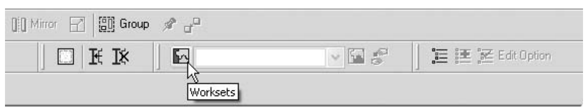


Figure 13.3 Accessing the Worksets command via the icon on the Worksets toolbar

If you can't find this toolbar, it may be shut off. As you can see in Figure 13.4, you can display the toolbar by right-clicking in the gray area of the toolbar area or by right-clicking on any toolbar icon. This will bring up a dialog box where you can select Worksets to toggle on the toolbar. Check marks indicate that the toolbar is turned on. These same toggles can also be found in Window ➢ Toolbar.

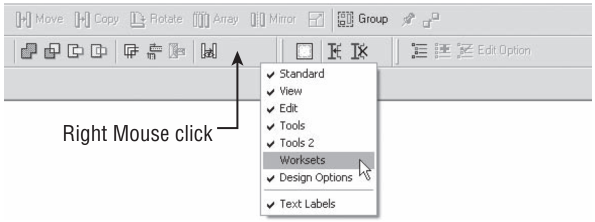


FIGURE 13.4
Right-click to display the Worksets toolbar.

With your single-user file open, click one of the Worksets buttons by using one of the methods we just described. You will be presented with a Worksharing dialog box (Figure 13.5).

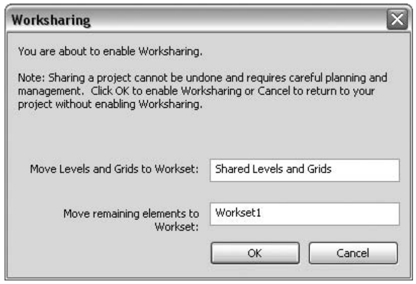


FIGURE 13.5
You enable worksharing in this dialog box.

This first dialog box warns you that sharing a project cannot be undone. This sounds pretty scary at first if this is your first exposure to worksharing. But don't worry; just proceed, and we'll explain what will happen.

During the brief seconds Revit Structure takes to enable worksharing, it places all elements inside the model into groups called worksets. User-defined worksets are created with the option to create additional ones, to give you a more hands-on approach of managing your model. The default worksets will be managed by Revit Structure as you work. To keep all of this in sync, worksharing makes use of both a central file and local files; the local files communicate back to the central file any ownership rights of elements.

User-Defined Worksets

In the Worksharing dialog box are two text fields available for editing. These are the two user-defined worksets that Revit Structure creates automatically when worksharing is enabled. You can create additional user-defined worksets after you click OK in the Worksharing dialog box.

SHARED LEVELS AND GRIDS

Revit Structure will take all currently modeled levels and grids and place them in a Shared Levels and Grids workset. Future levels and grids that you put into your project should be placed in this workset. You are allowed to place these elements into any user-defined workset, so if your project setup requires a different location, you can specify that.

Workset1

Everything that is not a level or grid will automatically be placed in Workset1. You should at least rename the default *Workset1* to something like *Main Workset*. This can be done right at the Worksharing dialog box shown in Figure 13.5, or you can click the Rename button in the Worksets dialog box. You can name the workset whatever makes most sense to you. The objects that are placed on this workset can be moved to another workset at a later time.

> **Amazing What a Name Can Do**
>
> Feel free to rename the default Workset1 to something more in line with your project, or even just *Structure*. Since all forms of Revit Structure applications use the same core programming, using more distinct names can help in subtle and unforeseen ways.

Explaining Default Worksets

After you click OK in the Worksharing dialog box, Revit Structure takes a few seconds to place all of your modeled elements into the two user-defined worksets. It takes all of your views, families, and project standards and puts them into an additional set of default worksets. Like the two user-defined worksets, these three worksets will be created automatically. Since these are hardcoded into the program, you will not be able to rename them.

Once Revit Structure has divided the project into its elected worksets, you will see the Worksets dialog box shown in Figure 13.6. Note that when you first enable worksharing you automatically become the owner of every workset—the Yes in the Editable column indicates this. You will also notice check boxes at the bottom of the dialog box. These check boxes allow you to toggle between the Views, Families, and Project Standards default worksets. And your current active workset is set to the user-defined workset Main Workset.

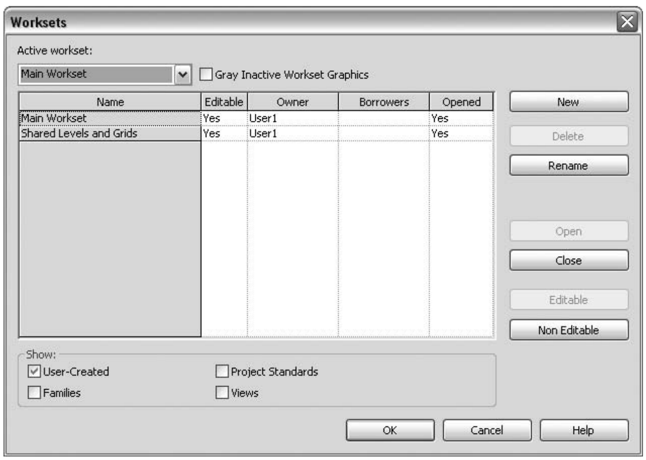


Figure 13.6
The Worksets dialog box

To create a new user-defined workset, click the New button in the Worksets dialog box. This opens the New Workset dialog box (Figure 13.7), where you name the workset. If you do not want to have this new workset visible in all views, deselect the Visible by Default in All Views check box.

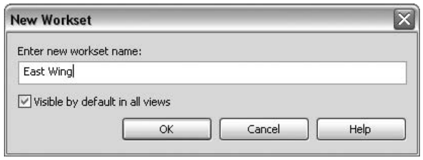
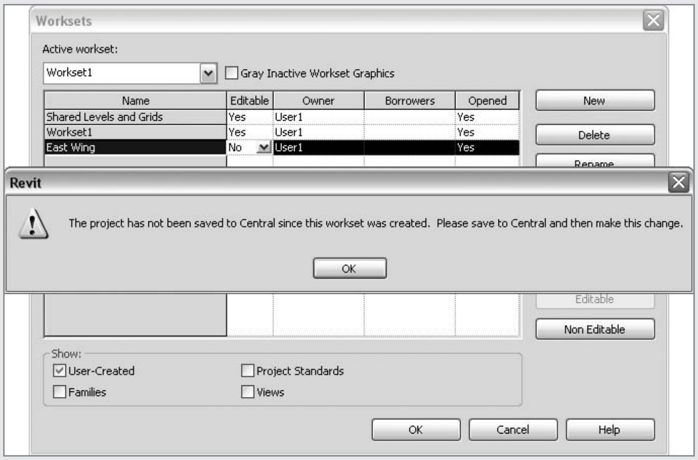


FIGURE 13.7
Creating a new user-defined workset

HOLD ON—YOU NEED TO SAVE TO CENTRAL FIRST

You will not be able to release your ownership of the worksets at this time. Setting the Editable column to No will prompt you to save the file first.



VIEWS

Revit Structure will place every view created inside your project in its own workset. This workset contains any view-specific elements such as annotations, as well as any view properties within the view. As you work in different views, these elements will automatically be put in the view's workset, regardless of which workset is current. Since Revit Structure controls all of this information, these worksets as well as the other default ones cannot be made active, nor can the elements be moved to another workset.

The nice thing about the View workset is that each user can have his or her own view of the model and control the properties. These properties will not affect other users who are in different views looking at the same portion of the model.

View worksets can be made editable in several places. You can right-click on the view name in the Project Browser or right-click on any view-specific elements to open a dialog box that lets you select Make Workset Editable. A third method is to go directly through the Workset dialog box.

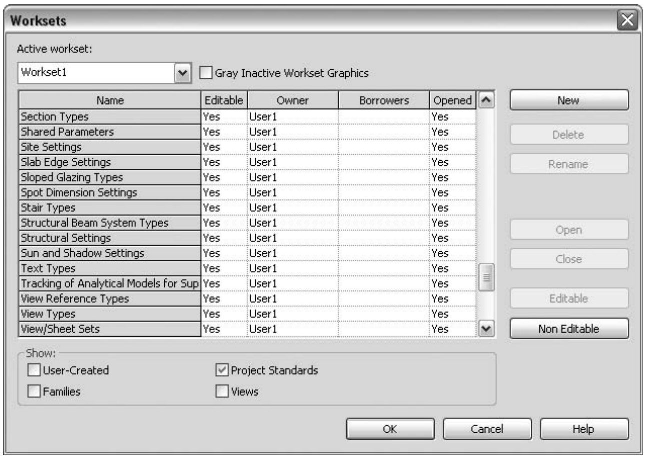
Families

Revit Structure puts every family that is loaded into your project in its own workset, which is automatically created for you. You will take ownership of any family that you edit the properties of.

Project Standards

All project settings are assigned to the Project Standards workset. Some of these settings include line styles, line weights, object styles, and text types. As Figure 13.8 shows, you can see all of them by clicking on the appropriate check box at the bottom of the Worksets dialog box. You do not need to take ownership of these worksets to add new information or additional settings to your project. If existing settings are modified, however, you have to take ownership of them.

Figure 13.8
Partial list of project standards worksets



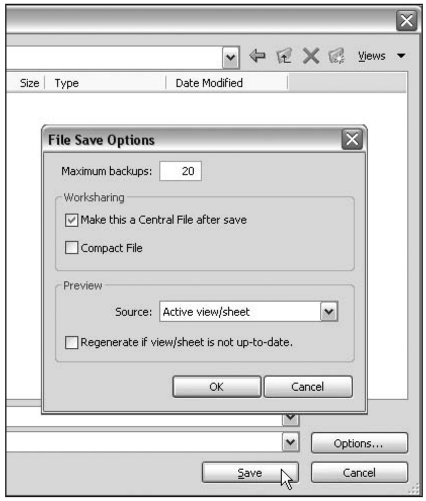
Creating the Central File

The first thing that you will want to do after enabling worksharing in your project is save the file to the network disk. This will become your central file, which will allow multiple users to then make copies of it for worksharing. When making your central file, it is a good idea to perform a Save As with a different filename instead of a Save. This will allow you to keep your original single-user file just in case you need it in the short term.

To create your central file, select File ➢ Save As. Before clicking the Save button, click the Options button in the lower-right corner. In the Options dialog box shown in Figure 13.9, verify

that Make This a Central File After Save is checked and the number of backups specified. The default is 20, which might not make your IT department too happy. Some large projects' backup folders can get quite large. You should discuss with your IT staff what your needs are and then agree on a setting.

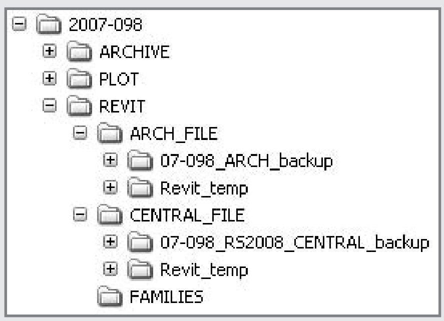
Figure 13.9
File save options



We All Need File-Naming Standards

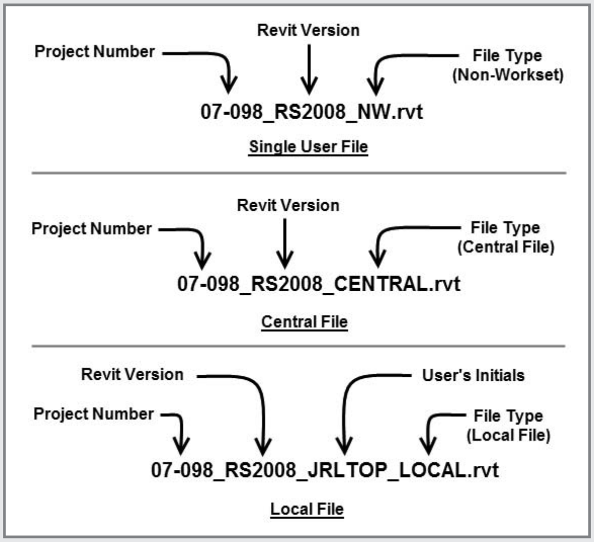
You should come up with a folder and file-naming convention for your central and local files that best suits your company. Try to avoid using special characters in your naming conventions. These characters can sometimes cause other applications to misbehave. If special characters are desired, you should check with your IT department to make sure your choice is acceptable to them.

On our server, we create a Revit folder inside our normal Project folder. This Revit folder holds all of our Revit–specific information for the project and keeps it apart from any AutoCAD files we may have during our transition to Revit Structure.



On each user's hard drive, we create a Revit_Local folder on the root of the C: \drive. Inside this we create individual project folders, where we save our local files and any other information about the project that does not need to be backed up on the network. Once again, we put the local file in its own folder to separate it from other project file information.

We use a file-naming convention that allows us to tie the project number to the filename as well as the version of Revit Structure that it is using. We specify whether it is a single-user, local, or central file. If it is a local file, the user will also include his or her initials.



After you create the central file, you will see that a couple of folders have been automatically created where the central file is saved (see Figure 13.10). You will only see these on a workshared project. One of the folders is named the same as the central file, with `.backup` appended to it. This backup folder is where Revit Structure keeps track of all of your backups pertaining to the central file. It contains a lot of `.rws` and `.dat` files that it writes so Revit Structure can monitor permissions and ownership rights of those using local files. To preserve these backups and other critical files, users should not delete or rename these folders or files.

A Revit_Temp folder is also automatically created. This folder provides information for the Worksharing monitor tool that is used to help users communicate with one another while they work together on a project. The files that are automatically created in this folder should only be a couple of kilobytes or less in size, so they will not take up much space; they are automatically

deleted when users close out of the project. If this folder is deleted, Revit Structure will automatically re-create it when any user resumes work.

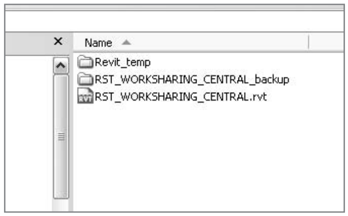


FIGURE 13.10 Additional folders are created in the same folder as the central file to store backup information.

Creating the Local File

There are two ways that you can make your local file. One way is to use Save As from within the central file, but this is not the preferred method for a good reason.

Opening the central file prevents other users from accessing it, which can prevent them from saving their changes back to the central file. You may only have it open for a minute or so, but it can be too easy to be interrupted or distracted by a phone call and continue working while still in the central file by mistake. With several team members working on a project and with this as your standard method, it is possible for more than one person to accidentally work in the central file. Users' normal interaction with the central file should be limited to working through the local file.

The preferred method is to copy the central file from the network to the user's workstation by using Window Explorer and rename it to your company's standard naming convention. When you do this, Revit Structure will automatically assume that the copy is a local file to the central file. You will see a dialog box when opening the file for the first time (Figure 13.11) explaining that the file will be a local copy and if you intended it to be a central file, you'll have to resave it as a central file.

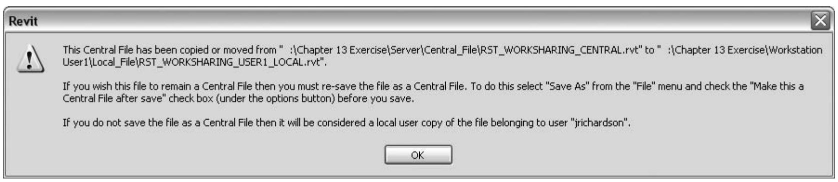


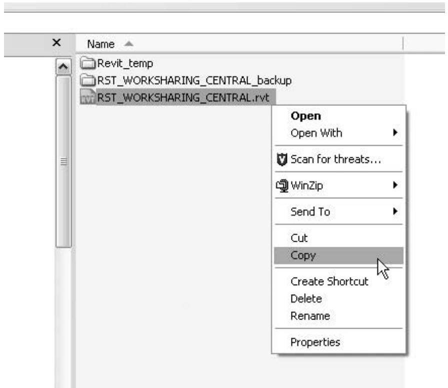
FIGURE 13.11 The warning dialog box

In some cases, users may already have the central file open—for example, right after you enable worksharing and create your central file for the first time. Even though it is much more convenient to create your local file by using the Save As method at this time, you should still avoid doing so. There is no sense in creating bad user habits.

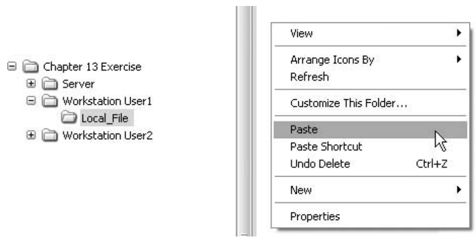
Let's step through the copy/paste method by using Windows Explorer:

1. Browse to your network where the central file is saved.

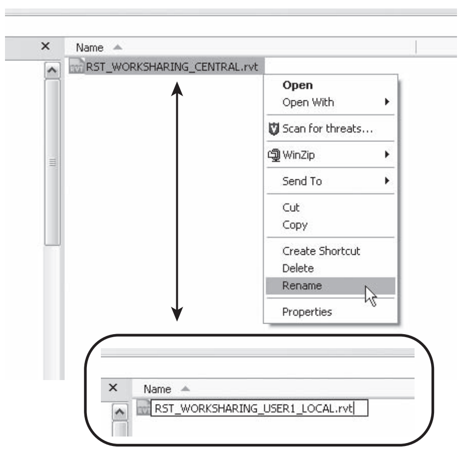
2. Right-click on the central file and click Copy.



3. Browse to the user's workstation's hard drive where the local file will be saved.
4. Right-click within the folder and select Paste.



5. Right-click on the pasted file and rename using the proper naming convention for local files.



6. Launch Revit Structure and open the local file you just created to continue working on the project.

How Can I Speed Up This Process?

Many companies have tapped into the API functionality of Revit and have created little programs that will automate the creation of the local file. They have also used batch routines or scripts that copy and rename the files. This allows the user to easily click on a shortcut or use a GUI interface to create the local file on a daily basis. This method enables company standards to be maintained and helps eliminate errors during the copy and paste process. A quick search on the Internet regarding scripts, Revit Structure, and local files should yield some examples or even full working applications you can put to use in your environment that have been created by users just like you.

Staying in Sync

Revit Structure has several commands to help you stay in sync with other team members while working together. Prior to enabling worksets, these commands will be grayed out. The majority of them can be found by selecting:

- File ➢ Save to Central
- Save to Central icon
- File ➢ Reload Latest

These are the commands that you will use to help the local and central files stay in sync with each other. It is good practice when working with worksets to save locally and save to central often. The more often you perform these saves, the less time it will take to update your model with the central file.

Saving to Central

Whenever you want to share your changes with the rest of the team, you will need to do a Save to Central (STC). This can be done in a couple of ways: by accessing the File menu or clicking the icon on the toolbar shown in Figure 13.12. There is a difference between the two.

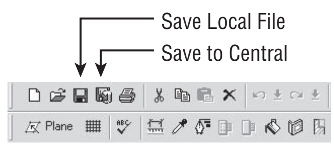


Figure 13.12
You can perform a quick STC by using the toolbar icon.

Clicking the icon will save your most recent changes to the central file since your last save to the central file and reload any changes since your last save back into your local file. Note that it does not automatically save your local file, nor does it return any user-created worksets that you may own.

The Save to Central option on the File menu opens a dialog box (Figure 13.13) as well as many more options for you prior to saving. You can choose to relinquish editable worksets, as well as save your local file after a Save to Central. User-created worksets will not be checked by default. You are also given the option to enter comments about your save. These comments can be used at a later time if you are looking at the backup history information. Using this method at the end of the working day ensures that all borrowed elements will be returned for other users to use and that both your local file and the central file are in sync.

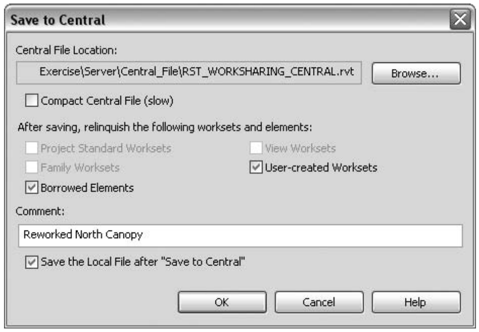


FIGURE 13.13
The Save to Central dialog box

Reloading Latest

There may be times that you need to get only the latest information from another team member. After this team member has saved his changes to the central file, you can then choose File ➢ Reload Latest and bring in only those changes. This command is nice to use when you want to see another user's changes but are not ready to publish your changes for others to see.

Working with Worksharing

While working in the worksharing environment using worksets, you will be presented with several new commands and notifications that help you stay in sync and share the project with other users. The central and local files along with their backup folders bring all this stuff together, which becomes a critical piece of the puzzle for allowing this type of environment to exist.

Making sure everyone knows the proper methods for working in a workshared environment and when to use these methods will help keep things assigned to the proper worksets. Even then, you will still need to move elements from one workset to another. Worksets help keep all team members working in sync while keeping track of who has worked on what with ownership rights. Above and beyond the assignment of ownership to elements you will find an opportunity to improve performance by controlling the display of elements assigned to these worksets.

Moving Elements Between Worksets

Once worksets are created, you will eventually have to move elements from one workset to another. This is usually done when you are moving elements from the default created Workset1

to another workset that you have created or moving additionally added levels in grids back to the Shared Levels and Grids Workset. It is also fairly common for users to place elements in the wrong workset by mistake as they are working. It's human nature, so you will more than likely have to perform the task of moving elements between worksets on a regular basis.

It is much easier to move elements between worksets when other users do not have elements borrowed or worksets checked out. It is good practice to communicate with other team members when moving large numbers of elements from one workset to another to avoid ownership conflicts. As Figure 13.14 shows, all elements will have additional Workset and Edited By parameters added to their properties under Identity Data when worksharing is enabled. The Workset parameter is what you will use to move elements between worksets.

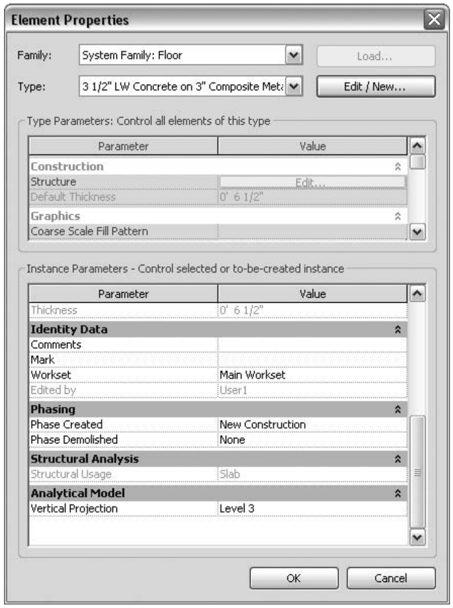
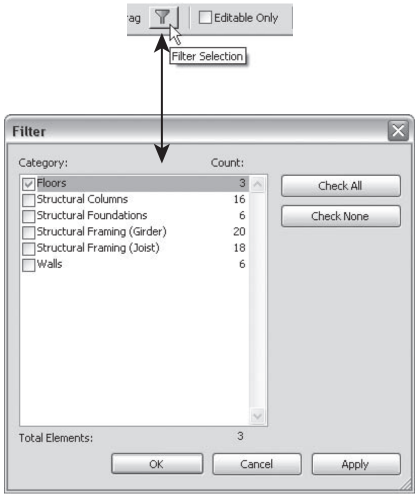


FIGURE 13.14
Changing worksets in the Element Properties dialog box

You can move elements to a different workset by selecting one or several elements at a time. If several elements of different categories are selected, the Workset Identity Data may not be available. Elements such as openings that become part of your selection can prevent the Workset Identity Data from being editable. For example, a workset that a wall opening is in is governed by the wall that it is placed in; if you change the wall workset, you'll see that the opening workset will automatically updated with it. Therefore, the wall opening would cause the Workset Identity Data from being editable.

A quick method to resolve this is to select several elements and use the Filter option on the Options bar to select only elements of the same category or those categories that will collectively allow for the workset identity to be changed. Figure 13.15 indicates that only floors will be selected.

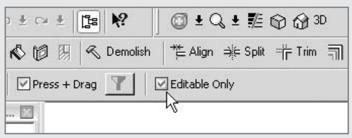
Figure 13.15
Selecting objects with a filter



If the Workset Identity Data is unavailable (grayed out), this may also be from you not having ownership of the element(s). You must take ownership before you can assign the element to a different workset. Elements assigned to the user-created worksets are the only ones that you can take ownership of; elements on default worksets will remained grayed out.

OK…Who Is Playing a Trick on Me?

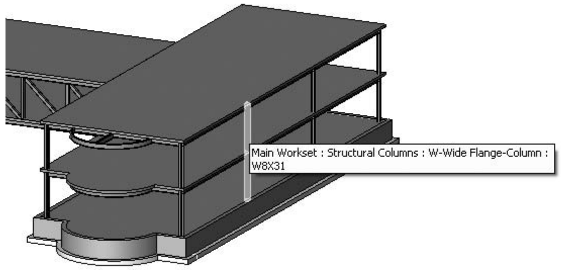
Suppose you're trying to select elements in a worksharing environment and no matter how hard you try you can't select them—you click but nothing happens. Make sure that the Editable Only box in the Options bar is not checked.



If this box is checked, you will only be allowed to select elements that you have ownership of. This might be why you cannot select the elements.

A quick way to verify which workset an element is assigned to is by hovering over the element with your cursor until an information box pops up. Not only does this box show you the element's category, family, and type name, but it will also show you the workset it is assigned to. In Figure 13.16, you can see that the element is part of the Main workset.

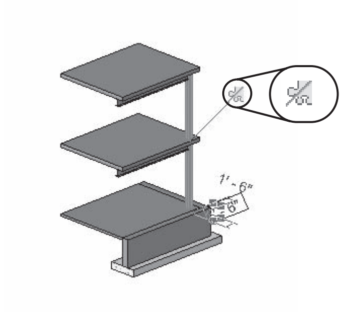
Figure 13.16
Hovering over an element to see its workset name



Taking Ownership

There may be a time where you will need to know whether you own a particular element. Figure 13.17 shows that if you do not own an element, you will see a puzzle piece–like icon (blue by default) when an element is selected. This is the Make Element Editable icon. If you select an element and you do not see the icon, you have ownership and can move or modify that element however you want. This icon is only available in a worksharing environment.

Figure 13.17
The icon indicates that you do not have ownership. Select it to take ownership.



The idea of borrowing elements versus checking out worksets is how Revit Structure allows you to work simultaneously with other users. Keeping this in mind, is important for all local files to maintain communication between the central file so that users can make editing requests and respond accordingly.

Borrowing Elements

By borrowing elements, you are only taking certain elements that are part of a workset. This will be your most common method of working, which allows you to only take ownership of the elements that you are working with. For instance, if a user has ownership of the East Wing workset and you need to make a framing connection and rotate one of the columns that is a part of that workset, you can make a request to only borrow that column. If the user who owns that workset grants you permission, you are allowed to borrow it. Once you are done making your change, you relinquish your changes.

Revit Structure will automate the process of borrowing elements for you as you modify them. You can also right-click an element and select Make Elements Editable from the context menu or select the little puzzle piece icon itself, as shown in Figure 13.18. This will allow you to borrow an element even though you might not be modifying it.

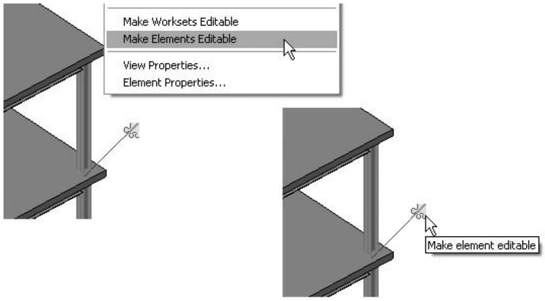


FIGURE 13.18
Methods of making elements editable

When you are at a point where you are ready to give up all of your ownership rights, you can do so by selecting File ➢ Relinquish All Mine. This will give up all ownership rights that you currently have without saving your changes back to the central file.

Checking Out Worksets

Checking out worksets is something you shouldn't have to do that often unless you are working with worksets that have entire portions of the building assigned to them. This is a bit different than borrowing elements because you take ownership of every element. Misuse of checking out worksets can prevent other users from being able to work efficiently.

You can check out worksets by opening the Worksets dialog box shown in Figure 13.19. Click on the workset name at the left and then click the Editable button at the lower right. A Yes indicates that it is editable by you and a No means it is not. You will also see whether you are the owner of the workset. If your username shows up under the Borrowers column, it means that you have elements borrowed from the workset.

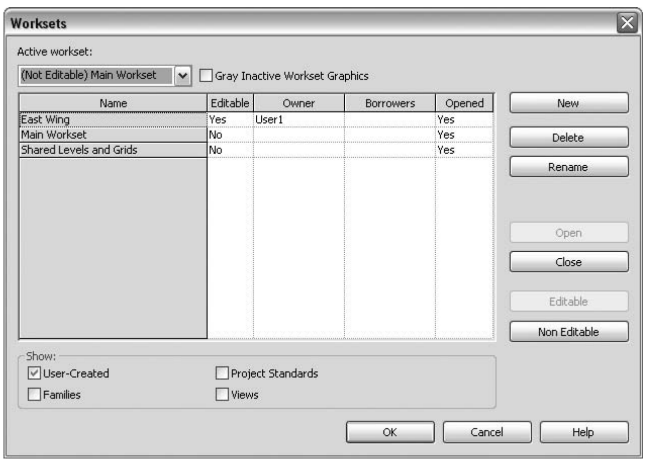
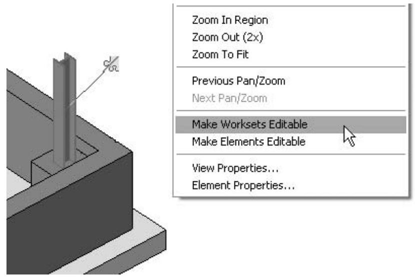


FIGURE 13.19
Checking out a workset in the Worksets dialog box

Figure 13.20 shows that you can also right-click on a selected element to select Make Worksets Editable. This will check out the workset that the selected element is on.

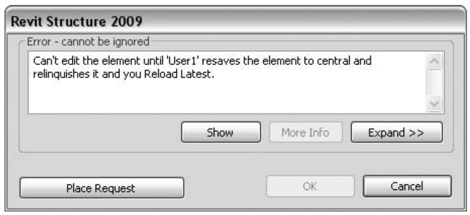
FIGURE 13.20
Checking out a workset through element selection



REQUESTING PERMISSION

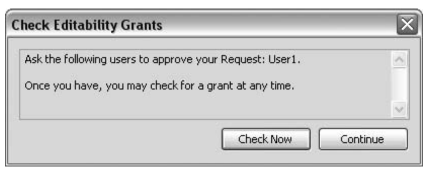
If you want to edit an element that another user owns, you can request permission to borrow it. At the same time, other users can request to borrow elements that you may own. If you come across an element that someone owns and try to modify it, you will be presented with the dialog box shown in Figure 13.21. This will only occur if you are working in a project where worksharing is enabled. This dialog box indicates that another user has ownership of the object. You can click the Expand button to get a more in-depth list of the errors and warnings.

FIGURE 13.21
This warning dialog box tells you that that another user has ownership of the object.



At this point, you have the option to cancel out and move on to something else or place a request for ownership of the element. Clicking the Place Request button opens the dialog box shown in Figure 13.22, where you can continue checking to see if your request has been granted. Clicking the Check Now button will keep looping back to the same dialog box until the other user has taken action. If you do not want to wait, you can click Continue then Cancel out of the remainder dialog box. This will not cancel your request but will allow you to continue working on another area until your request has been granted.

FIGURE 13.22
The Check Editability Grants dialog box



Once you place the request, you have to wait until the current owner grants you permission. Other users will not automatically receive any indication that you have made a request for an element that they own. You need to physically make contact with the user by calling, emailing, tapping on the shoulder, or by using the Worksharing Monitor. (The Worksharing Monitor tool is discussed later in this chapter.)

Granting Permissions

Once you have the user's attention, they will need to go to their editing request window located in File ➢ Editing Requests, or as Figure 13.23 shows, click the Editing Requests icon located in the Worksets toolbar.

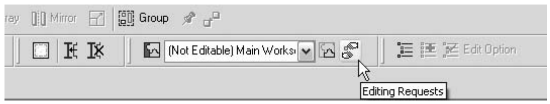


Figure 13.23
Accessing the Editing Requests command

The Editing Requests dialog box will show you any request that have been made to you as well as any pending request(s) that you may have. Figures 13.24 and 13.25 show two different Editing Requests dialog boxes. Three options are available to address the request:

Grant This option gives permission to the user making the request.

Deny/Retract This option denies permission to the user making the request or allows you to retract your request to another user.

Show This option shows you the exact element(s) that have been requested.

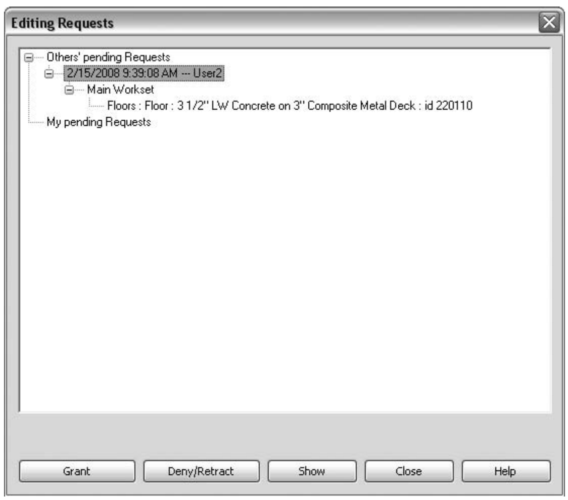
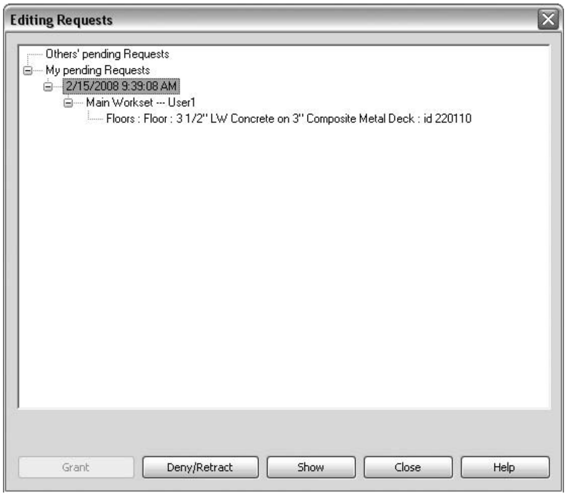


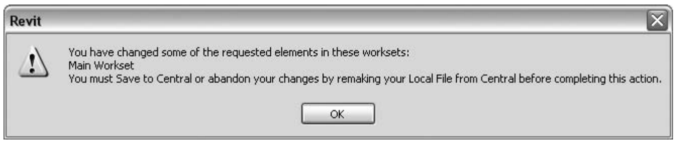
Figure 13.24
Others' pending requests

Figure 13.25
Your pending requests



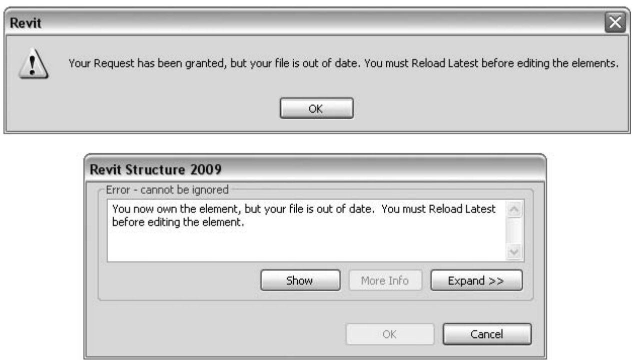
In the case where a (*) is shown, as in Figure 13.24, you will need to Save to Central before granting the request. Granting without doing this will present you the dialog box shown in Figure 13.26. This occurs because the central file does not have the current information that your local file has. You will need to exit the dialog box and Save to Central before granting the request.

Figure 13.26
You'll see a warning when your local file is not in sync with the central file.



After the request has been granted, the user who is making the request will see one of the two dialog boxes shown in Figure 13.27. The ownership has been transferred over, but the local file is out of date, so a Reload Latest or a Save to Central will have to be performed before editing of the element can occur.

Figure 13.27
Warning dialogs stating that your Local File is out of date



Working at Risk

You may have to work on your project when the central file is not accessible for the local file to communicate through the bidirectional link. If this is the case, you will see the dialog box shown in Figure 13.28, telling you that your connection cannot be found and your changes may be lost. This basically means that you are working at risk.

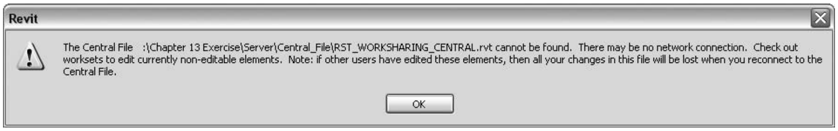


Figure 13.28 The central file cannot be found.

Revit Structure has no way of communicating back to the central file any ownership information that you may have created. If another user working on the project takes ownership of elements or worksets that you borrowed at risk, you will be prevented from saving to the central file, thus losing your work. Your work will be lost, although you will be able to copy elements from one model to the other.

If you are working at risk and try to check out a workset, you will see a dialog box similar to Figure 13.29 warning you yet again that you are in an at-risk worksharing environment.

Figure 13.29 Editing at risk warning

You should use caution when using this procedure. If you do decide to work at risk on your project, communicate your actions with other team members.

Improving Performance

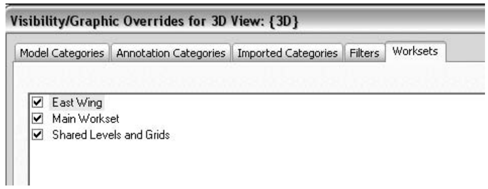
It is pretty easy to go overboard with detail in your project. The natural tendency is to model everything down to each and every little plate and connection. When doing this, the size of your file will increase significantly and will slow down the worksharing process. Keep this in mind when modeling in your project.

Take advantage of the additional options that become available to you when worksharing is enabled by shutting off the display of worksets in certain views, or leave them closed with a Selective Open operation when opening your project.

Using Visibility Control

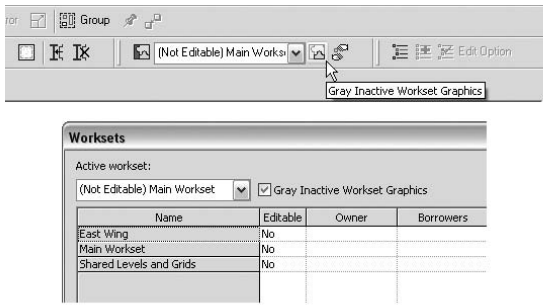
With worksharing enabled, an additional tab called Worksets becomes available in the Visibility Graphics Overrides dialog box. As Figure 13.30 shows, you can toggle the visibility of any user-defined worksets on or off. Access this dialog box by right-clicking in the view, selecting View ➢ Visibility Graphics, or using the command Aliases VG /VV. On the Worksets tab, you can check the boxes to show the worksets or deselect the boxes to hide the worksets.

FIGURE 13.30
Controlling the visibility of worksets



Also available is an option called Gray Inactive Workset Graphics, which can be activated from different locations: by clicking the icon on the Worksets toolbar or by clicking the checkbox in the Worksets dialog box (see Figure 13.31).

FIGURE 13.31
Activating the Gray Inactive Workset Graphics option



Making use of the Visibility Control allows you to easily turn off worksets that are not needed in an individual view for display or just not needed at the time that you are working in that view. Since several categories can be assigned to a workset, it is much easier to toggle the workset on and off, rather than picking individual categories. If the toggle of display is done as a temporary override, make sure that you toggle it back to its original state because it will not reset itself. The Gray Inactive Workset Graphics command can be used to visually show you elements that are not in the current workset or to display the elements you might not be working with but want to see in the background as a nonemphasized element. The nice thing about this command is that if you forget to toggle it back to normal and you plot, line weights will still plot correctly.

SELECTIVE OPENING OF WORKSETS

There may be times when you do not need to see particular worksets while you are working. For instance, if you have a workset called East Wing and you are not working in that area, not opening that workset will increase your performance because those elements do not have to be displayed. When opening your local file, you can access the open options shown in Figure 13.32 by clicking on the down arrow next to the Open button.

- All opens all the worksets in the project.
- Editable opens all worksets checked out by you.

- Last Viewed opens the worksets that were opened when the project was last closed.
- Specify opens the workset selected from a list in the Opening Worksets dialog box.

Linking other discipline models like architectural, mechanical, or civil into your project and having them all loaded to display at the same time can consume your computer resources. Placing each of these links in its own workset will allow you to only load the ones in that you need. If you will not be working with coordinating mechanical for the day, you can choose to not open that workset. This can be a much better approach than reloading and unloading the link in Manage Links. Making use of the Performance Monitor in the Worksharing Monitor will help you determine the demand these extra linked files can put on your system's resources.

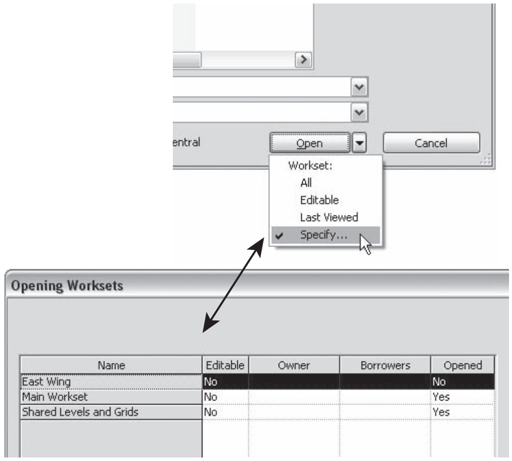


FIGURE 13.32 The Open commands for worksets

Using the Worksharing Monitor

The Worksharing Monitor is only useful if you are working on a project with worksets enabled. The only use for a single-user model environment is to display your current system's performance. This application is not part of Revit Structure's installation. It is a separate extension available through Autodesk and is strictly a tool that monitors temporary files that are placed in the Revit_Temp folder adjacent to your central file.

So, how can the Worksharing Monitor help you in your workflows? Well, it solves many questions that you might have as multiple users work on your project. For instance:

- How long has it been since your last Save to Central?
- Is your local file out of date?
- Other than yourself, who is working in the project?
- Is someone working in the central file?
- Has your request been granted yet?
- Who last worked on the central file?
- Do you need to respond to any editing requests?

Once the Worksharing Monitor is installed, you will need to launch it separately from Revit Structure. You can do this by clicking on the shortcut that Revit Structure places on your desktop, going through the Windows Start menu, or (within Revit Structure) by choosing Tools ➢ External Tools ➢ Worksharing Monitor.

Since this is a separate program and has its own window, it is nice to have a dual monitor setup. However, if you do not have dual monitors in your environment, you can configure the Worksharing Monitor to display desktop alerts. These alerts will show up in the bottom-right corner of your screen with brief messages. At that point you can bring up the Monitor window for additional information. Keep in mind that this is only a monitoring tool and all actions by you still need to be done from within Revit Structure.

When All Else Fails, Call for Help

Your best source of information regarding the Worksharing Monitor is the help documentation in the Worksharing Monitor program. This documentation gives an in-depth look at how the tool works and looks. It also features several graphics and explanations of common day-to-day scenarios.

Using the interface of the Worksharing Monitor is quite simple; it allows you to set it to interact with a user's individual work methods and preferences. It displays the pertinent information needed at your fingertips while you are working.

Observing the Graphical User Interface (GUI)

The Worksharing Monitor's main display shown in Figure 13.33 is organized into three sections:

Central File Access This section shows information about the project you are working on, such as the central and local filenames as well as who is working in what files. It also will tell you when other users are saving and how long before they are finished saving.

Editing Request This section displays any editing request that you may have made of other users. It displays the current status as well as who created the requests and who needs to respond.

Notification This section displays warning information such as low memory, potential delays in saving, or users not informed of your request.

The Options button allows you to configure the notifications for behavior of each section in the main dialog. You have control whether or not you receive pop up messages and if so how long do they display before disappearing.

The System Performance Monitor shown in Figure 13.34 is a great little tool for showing you your system's performance when you are working on your project. Since this dialog pulls information from your system rather than the Revit_Temp folder, this feature will work when working on a single user file.

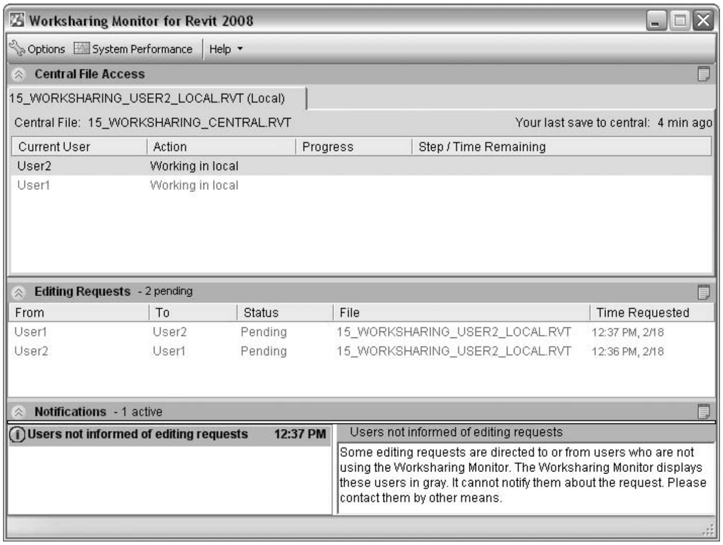


FIGURE 13.33 The main GUI screen for the Worksharing Monitor

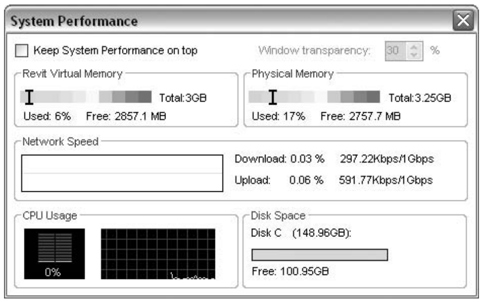


FIGURE 13.34 The System Performance Monitor from the Worksharing Monitor keeps track of your computer's resources

Maintaining Your Project

It is good practice to do regular maintenance on your central file. This includes keeping all users informed of how they should be working with the central and local files as well as what to do when things go wrong. Most of this maintenance should be done by a team captain or project manager so proper notification is given and tasks are performed consistently.

There are times when you will need to access the backup files for your projects. Most of the time, this happens in the heat of the moment, so knowing how to do this, and feeling comfortable doing it, is something you will have to develop. Teaching users when to use the Detach from Central feature might help avoid trouble. Proper upgrading and routine audits will help keep your central file healthy and free of corruption.

Accessing Backup Files

Salvaging a workshared project's backup file is a little different than just renaming the BAK file as you did in AutoCAD. You can use a command that is located in the File menu: Backups.

After selecting File ➢ Backups, browse to the backup folder that you want to retrieve (roll back) backups from. Backups can be retrieved from either the local file backup or the central file backup. The number of backups you chose to keep when you created your central and focal files determines how far back your backups will go.

Once you have located and opened the backup folder, you will be presented with a list of backup files similar to Figure 13.35 that are available for a Save As or Rollback operation. The number of backups is specified in the Options button when you perform a Save As. The default is 20; in Figure 13.35 it is set to 4. This is where it can become useful to add comments when you Save to Central. It will allow you easily determine which backup you need to get back to.

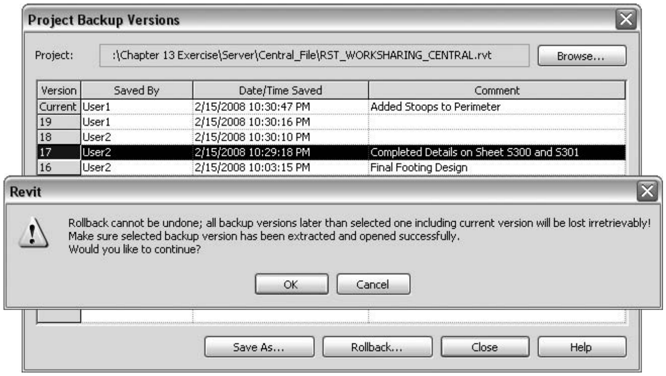
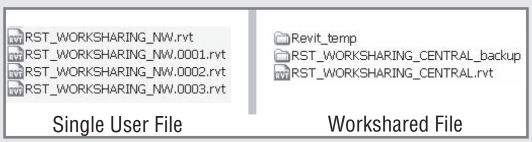


FIGURE 13.35
Salvaging a backup from a workshared project

Before attempting to roll back a project, take a few minutes to think about what will be happening. Most of the time when you have to perform this action, you are in the heat of the moment and not always thinking clearly. All later versions in the backup folder will be lost when you roll back a project, so you might want to perform a Save As to save the backup to a new project first.

WHAT IS IT, A SINGLE-USER OR WORKSHARED FILE?

Determining what type of file your project is can be pretty simple. If your project file has additional files adjacent to it with `.000X.rvt` appended to the filenames, then it is a single-user file. These files can be renamed (remove the `.000X`) just as you would rename an AutoCAD BAK file.



If your project file has an additional folder next to it with the same name and backup appended to it, then this is a workshared file. You need to use the Backup command to retrieve backups.

Detaching from Central

If for any reason you need to work inside the project and you know that you do not want to save your changes or risk borrowing elements that you do not want to, you can perform a Detach from Central prior to opening. This option is quite useful when you're performing tasks that do not need to be saved to the central file, such as:

- Transferring project standards
- Archiving a copy of the project
- Reviewing the project (by a manager)
- Exporting views for client collaboration
- Displaying the project in meetings
- Performing studies

Detach from Central is available in the Open dialog box. When you select the Detach from Central check box, you will see the notification shown in Figure 13.36 telling you that you are creating an independent file. All path and permission information is reset and no changes can be saved back to the central file. If saved, the file will become a new central file. If this check box is selected for a single-user file, the option will be ignored and the file will open normally.

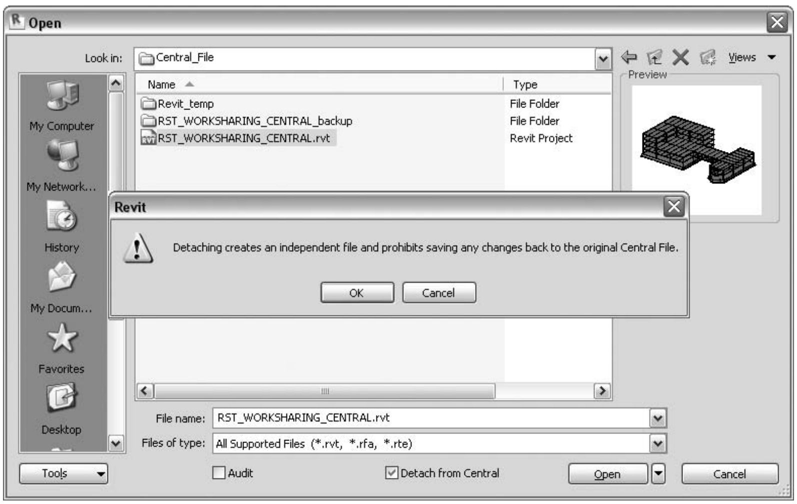


FIGURE 13.36 Detaching from Central to make an independent file

Upgrading the Central File

When upgrading to a new release of Revit Structure, you should use the following procedure to upgrade the central file for each of your projects using worksharing. Create a backup copy of your project in the current release before performing the upgrade. This backup file will be valuable if for any reason the new central file becomes corrupted.

To create your backup copy, make sure that all team members have saved their latest information to the central file. With the current release, open up the central file by using the Detach

from Central option. This will ensure that all user ownerships are relinquished and allow you to create a clean central file with a new name.

To retain your backup history, you will have to copy the central file and its backup folder to a new location. Be careful because the copy is actually making a local copy.

When upgrading the project, make sure that any Revit links in your project are upgraded first. To upgrade your file, open in the new release of Revit Structure. You will see the dialog box shown in Figure 13.37, indicating that your file is being upgraded. To ensure that the backup history of the project is retained, perform a Save to Central from the File menu in lieu of using the Save As command.

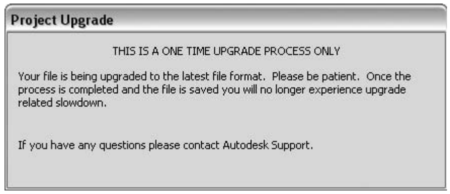


FIGURE 13.37
The Project Upgrade dialog box

Auditing Your Project File

Part of keeping your central file healthy is occasionally performing an audit on it. This should be done at least once a month, and you definitely want to perform an audit before you upgrade to a new release. This will help fix and detect any corruption that might be inside the database. The Audit check box appears at the bottom of the Open dialog box, shown in Figure 13.38. Performing an audit on your project can increase the time it will take to open your file. Keep this in mind on some of your larger projects. Prior to performing this task, tell all users to save their local files to the central file and close out of the project. After auditing, have all users create new local files.

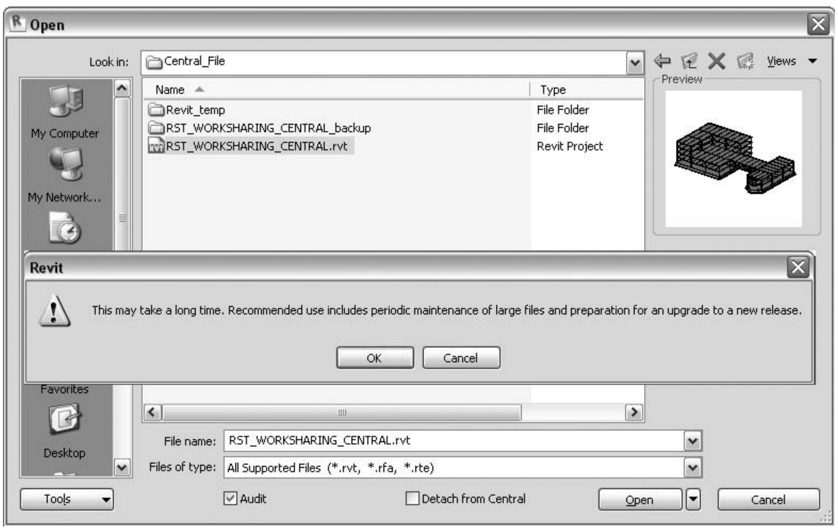
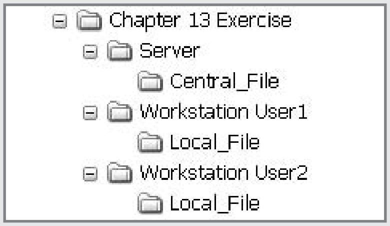


FIGURE 13.38
Audit your project file before upgrading to a new release.

Exercise: Working in a Multiuser Environment

For this exercise you are going to make temporary settings to your Revit Structure session(s) to simulate two users working in a worksharing environment. This will require you to open two sessions of Revit Structure on your workstation and reset the username of each session on the General tab located in the Settings ➢ Options menu. *Remember these settings because you will want to set them back when you are done with the exercise.*

Before beginning, you will need to use the `RST_WORKSHARING_NW.rvt` file (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`) and set up a folder structure like the one shown in the following illustration on your workstation's C:\ drive. This folder structure will act as both the server and users workstations. In a real-world scenario, the single-user file and the central file would be saved to the network.



Once the folder structure is created, copy the `RST_WORKSHARING_NW.rvt` file into the Server folder. This will be your single-user file saved to the server.

Enable Worksharing

1. Launch two sessions of Revit Structure on your workstation.
2. In session one, select Settings ➢ Options and change the username to **User1**. Repeat the same steps for session two but use **User2**. (From now on we will refer to User1 and User2.)
3. **User1:** Open `RST_WORKSHARING_NW.rvt` and take a look at the project. Notice that part of the Standard toolbar, the Save to Central icon, is grayed out and the Worksharing commands in the File menu are grayed out. This indicates that worksharing is not yet enabled.
4. **User1:** Enable worksharing.
5. **User1:** Change Workset 1 to **Main Workset** and click OK.
6. **User1:** Take a look at the Worksets dialog box to see how the existing elements are organized into worksets. Toggle through the check boxes for the default worksets. Do not exit the box yet.
7. **User1:** Create a new workset called **East Wing** and click OK to exit.

Create the Central File

1. **User1:** Save your new workset model to the server in the Central File folder as `RST_WORKSHARING_CENTRAL.rvt`.
2. **User1:** Choose File ➢ Save to Central to relinquish all worksets. Make sure all Workset boxes are checked prior to clicking OK.
3. Close out of the central file (File ➢ Close).

Create the Local file

1. Using Windows Explorer, copy the central file on the server to the Local_File folder on Workstation User1.
2. Rename the new local file to **`RST_WORKSHARING_USER1_LOCAL.rvt`**.
3. Repeat Steps 1 and 2 of this section for User2's local file.
4. **User1:** Open the local file `RST_WORKSHARING_USER1_LOCAL.rvt`. Review the "This is not a Central File…" dialog box and click OK. User1 is now working in a local file that has a two-way link with the central file.
5. **User2:** Repeat Step 4 of this section.

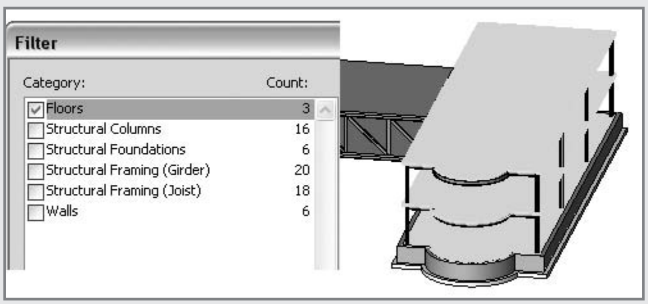
Review New Items

User1/User2: Observe the additional tools and commands that are available to you:

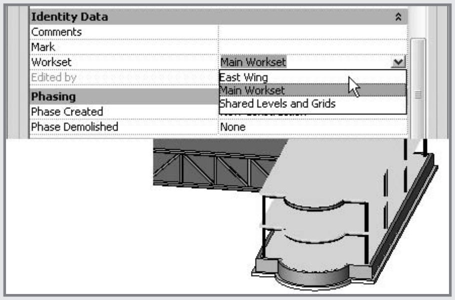
- Workset toolbar
- File menu
- Visibility Graphics Overrides
- Element/View Properties

Move Elements between Worksets

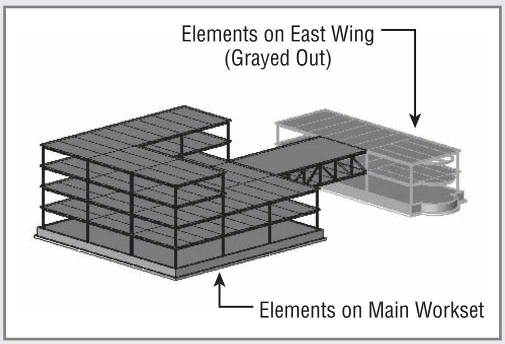
1. **User1/User2:** Choose File ➢ Save to Central. Click any required workset toggles and check the Save the Local File option to make sure you did not take ownership of any elements while observing the model.
2. **User1:** Take ownership (Check Out) of the Main Worksetworkset. You will be moving elements from this workset onto the East Wing workset.
3. **User1:** Go to a 3D view and orient your view so you can select the East Wing objects with a left-to-right crossing window, as shown here. Once the objects are selected, use the filter selection to only select elements of the same category.



4. **User1:** Open the properties of the selected elements shown here and find the Identity Data section. Change Main Workset to **East Wing**. Click OK.



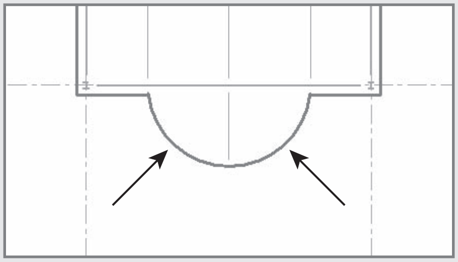
5. **User1:** Repeat Steps 3 and 4 of this section for all elements until they are all on the East Wing workset. Tip: Turn the East Wing workset off in the Visibility Graphics Override dialog box so the elements disappear when you put them in the East Wing workset. When you are all done, you can turn the workset back on to see your elements. Note that having to select elements of the same category may not always be the behavior that Revit Structure presents. Depending on the element types within your selection set you may be able to select several elements from several categories and still have the option to switch their workset to another workset without having to repeat steps 3 and 4.
6. **User1:** With the Main Workset active and East Wing workset turned back on, gray out the Inactive workset with the "Gray Inactive Workset Graphics" icon on the worksets toolbar. You should see the East Wing grayed out as shown here.



7. **User1:** Choose File ➢ Save to Central, and click workset boxes and the Save the Local File option to relinquish ownership of any elements.
8. **User2:** Choose File ➢ Reload Latest to update your local file with User1's recent changes to the East Wing.

PERFORM EDITING REQUEST

1. **User1:** Switch to the Level 3 view and edit the slab edge to run to the centerline of the curved beam as shown. When complete, finish the sketch.



2. **User2:** Switch to the Level 3 view and attempt to edit the slab edge. *Note that you do not see the new slab edge edits in User2 local. This is because User1 did not do a Save to Central.* Attempting to edit the slab edge at this point will prompt you to make a request because you do not own the element.
3. **User2:** Place the request and notify User1.
4. **User1:** Open the Editing Request dialog box and observe the message. In this case User1 will have to perform a Save to Central before being able to grant the request. Do the Save to Central, then grant the request.
5. **User2:** Check to see if your request has been granted. Yes, you should now own the element, but since your local file is out of date and does not reflect User2's recent slab changes, you will need to perform a Reload Latest before you can edit the elements.

This concludes the brief exercise for this chapter. Continue to work on the files on your own as if you were working in a multiuser environment. When you are done, make sure that you set your username back to its original setting.

The Bottom Line

Determine when to enable worksharing. Looking past day one of your project to help determine the proper game plan for moving forward will make things go much smoother.

Master It What can determine when you should enable worksharing?

Enable and set up the worksharing environment. When working in Revit Structure, you'll almost always be in a multiuser environment. Knowing how to enable worksets and use the central and local files is important for communicating between team members.

Master It How do you enable worksets and where should the central and local files be saved?

Request and grant permission of elements. Working in a multiuser environment where you are sharing a project with ownership rights will eventually lead to team members tripping over one another. Understanding how the ownership rules are set is important when working efficiently.

> **Master It** What are the methods used so you can take ownership of an element(s)? What do you do if another team member has ownership of an element(s) that you need?

Stay in sync with other team members. Creating the central file and communicating to it with a local file is how you effectively work in a workshared environment.

> **Master It** What is the recommended way to create a local file? What commands are used to get information back and forth between the central and local files?

Properly maintain your project file. Keeping your central file as healthy as possible will decrease chances of file corruption and increase overall performance.

> **Master It** When should you audit the central file? How should you go about upgrading your file to a new release?

Chapter 14

Visualization

You have just spent a significant amount of time learning about Revit Structure, creating models and documenting them. But once you have your *regular* work done, you should take some time to show it off. However simple you will find modeling your buildings, it may be difficult to gain a high level of familiarity with the rendering process because you so infrequently render your projects. But keep in mind that you can gain much by exploring and implementing the rendering features contained in Revit Structure.

The rendering engine in Revit Structure enables you to create visual imagery of your models. You can then leverage these images in everyday as well as specialized uses.

In this chapter, you will learn to:

- Determine what and when to model
- Assign materials to your model
- Define the quality and style of your renderings
- Export your models for other uses

How Much Do You Need?

When you begin to plan your models, undoubtedly you make sure you are covering all the basics. You will have grids, columns, floors, beams, walls, and so forth. You will then begin your documentation process, adding details and schedules as needed. Some will be generated live by the model, while others will be just lines and text placed alongside your model objects. This will continue during the entire sequence of construction drawing development. Soon you will have a nice—and probably bloated—model, ripe for using beyond normal drawings.

But will you be able to use the model for these extra purposes? If you modeled every piece of steel in a composite steel structure, not only would your model be huge and probably slow, but the level of detail in it might very well be beyond extended use. If you make a highly detailed model and it takes an hour to render it once, that probably will not be suitable for your rendering needs.

In the next section we will cover developing limits on what to model, along with methods to understand and control model bloating.

Modeling Beyond the Paper Documents

The normal process of modeling should be measured against several tiers. The first tier is your plan drawings. If you *see* something on plan and not in any other view, the need to model in 3D, to start with, is subjective. Sure, do it, if the object in question is easy to make and a primary component of Revit Structure. Then there are the items you see on plan and in sections or elevations. These too

should probably be modeled if for no other reason than you don't want to create them twice (plan and detail). It also will help that it is dynamic (all views will be automatically updated).

So what do you *not* model? For example, on a steel joist roof system bracing angles are often required for the structure design. These items could be modeled and then shown on plan—but it is doubtful that you have to model them since these small items would not normally be shown in a building section.

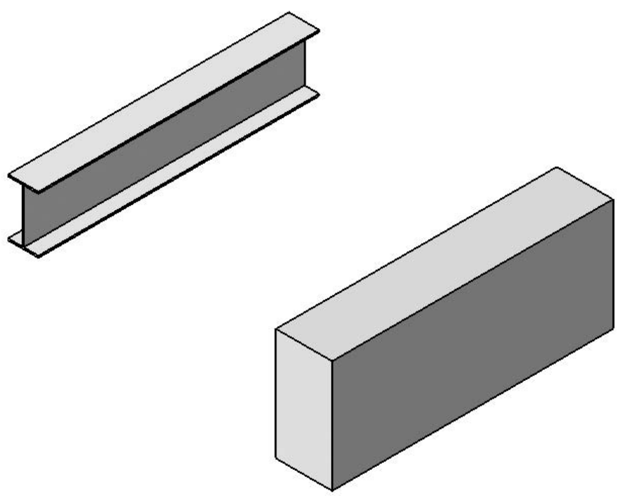
But let's imagine for a moment that your project has a healthy fee and you have the time to model every little thing. Not only will it be fun to create that virtual building, but you will get the added benefit of doing material takeoffs quickly. You can also use these components for the next project, where you may not have the luxury of time. You have no reason to not model every little thing. But you know there has *got* to be repercussions for that!

That "speed kills" is old adage that many relate to driving an automobile. Go too fast and you lose control. Well, excessive modeling is sort of the opposite; you will have no usability or speed and simply want to kill yourself waiting for the model to respond! Certainly this is a joke, but many times it has proven true. If you model too much, your model will become so slow that you can't work at any reasonable speed—and that is if you are lucky! Overly large models also can become unstable and crash without much warning.

Now what does this have to do with rendering? It's simple; the model you make will "make or break" your modeling efforts. If a primary goal of modeling in Revit Structure is to create a nice rendering view, you should keep that in mind while you create your model. However, if your project is small and simple, you cannot overmodel beyond the rendering capabilities. If your model is a convention center with tens of thousands of steel sticks, that may pose a problem.

As you have learned already, Revit Structure is a database-driven system. Every single thing in the model impacts its size. In Figure 14.1, we have a steel beam next to a concrete beam. Do you think these two are the same in the BIM world? In a way, yes, both are beams, but in every other way, no. The concrete has properties to help it *be* a concrete beam. Likewise, the steel beam has properties to help it *be* a steel beam. But due to the nature of this object type, there is a lot more information for a steel beam. Not only does it have connection symbols, but a steel beam has additional invisible data in this view; the stick line representation as well as the fine detail level information, which would show the curved interior edges of the wide flange.

FIGURE 14.1
A face comparison between a steel and concrete beam



When you activate the rendering process, the surface of the shapes is what takes Revit Structure time to process. A concrete beam with six faces (four sides and two ends) will take less time to render than a wide-flange beam. A wide-flange beam in general has 12 faces, but depending on the meshing/rendering process, it can have 18 faces. To figure it out for yourself, cover your shape with "rectangles" and then count them up. So it would be logical to think that a steel beam can have three times the face count of a concrete beam, and then perhaps take three times as long to render. A project with 5,000 steel beams would have a face count of 90,000—just for the beams. It is not unrealistic to have a project with over a million faces.

The key thing to keep in mind is to render what you need and not just what you have. If you can use a simplified shape, do so to cut down on the face count. Perform a regular basic rendering of your model, and track the model file size to compare with how long it takes to render. There is nothing worse than trying to get a high-quality render made but not have the time to create it.

Model Creep

Model creep isn't a term to describe a mean model or scary one, either. It is simply explaining a tendency to model beyond your needs, resulting in a bloated model. Imagine you have a team of people working on a project and everyone is actively contributing to the model. You should approach your virtual model just as the general contractor would. There would be regular meetings to discuss what is getting *built* today, what not to *build* at all, and what is still to be *built*. Your virtual construction should be no different, if for no other reason than to keep track of what goes into your model. You might be a frugal modeler, only putting in what needs to be put in. But your coworkers might not be so studious and go overboard. They're off modeling turndowns on slab-on-grades. They're modeling plates for steel beam to column connections, as shown in Figure 14.2. They're placing cold-formed steel stud members at 16″ on the center in interior walls. If these are things you need to model, then all is good. But if not, you and your team need to talk. Any valid additional items would have to be managed and/or filtered so that you can later turn them off for your renders if needed.

Real World Scenario

Understanding Client Needs

We were developing a rather large project in early design. The project manager asked that a detailed model be created with specifics for beam sizes and such. After a few days of model creation, the topic of renderings came up. "Can we produce some images to wow the client?" "Absolutely, no problem. Just tell me where you want to look from." "Okay, can do." Of course more modeling went on without a word about location for point of view. A week went by and the model was ready for rendering. Seeing the deadline approaching, typical assumptions are made as to where to "see" the model and renderings began—except they took forever! The model was so dense that each rendering took 90 minutes. It turned out okay—there was enough time to run the shots. But disappointment came when the renderings were handed over and it was decided a single shot was all that was needed. We could have modeled less than 30% since the model was never seen by the client, internal superiors and external customers.

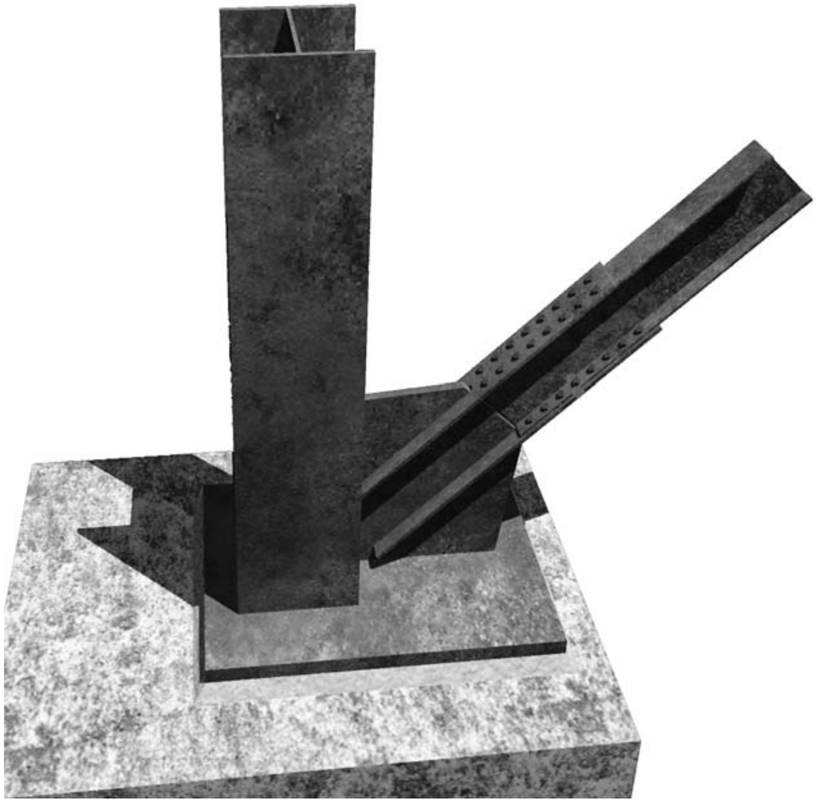


Figure 14.2
This is a nicely detailed connection, but modeled thousands of times it would bring rendering to a crawl.

The crux of creep is that you need to watch and be aware of what goes into your models so that you can better prepare for when you need to create renders. Once you know what to look for, you can begin to do some work!

Revit Structure mental ray Rendering

Whereas the previous Revit rendering engine, Accurender, was a part of the Revit product line for some time, now that Revit is owned and developed by Autodesk, it can take advantage of 3ds Max technology and its mental ray rendering engine. A key benefit is a more consistent render output across various software platforms. Another key benefit is that in Revit Structure a less intensive configuration process is required to produce desirable results.

Materials

Materials are the make-or-break component of a successful rendering. The world is your oyster, in a sense. You can make your rendering look like a cartoon, make it look realistic, or anywhere in between. The funny thing is that materials are often a source of so much aggravation, not because of the difficulty in using them but rather because everyone has an opinion, and just like art, beauty is in the eye of the beholder. Whereas lighting controls what you see and how flat or realistic the rendering is, materials will convey object type regardless of the lighting.

As you begin to create renderings on a regular basis, you should define whatever styles you will offer to your clients, then allow your clients to pick from your portfolio of options and the style of rendering they want you to develop. You might have a single color material (gray) that you use for all objects. This would convey, without question, that the rendering is a computer model. This technique is best in early stages when form and volume are most important.

Another method is to have materials of a single color representing the components on the structure—for example, a white/gray for concrete objects and a deep red for steel objects, representing primer paint. You can use a photograph of a concrete wall surface with form holes as well as stains. This would be well suited for close-up images where the detail of the surface is an added bonus for faking realism.

Materials in Revit Structure can be thought of in two distinct fashions: those that are faked and those that are made of something real. In the following section, we will cover synthetic and realistic materials.

Synthetic

The first form of material we'll refer to as *synthetic*. It does not exist in the real world and generally is just a color with various reflective properties, like a paint. A synthetic material is usually the best option for large-scale renderings. One caveat: Since a synthetic material itself has no patterning, you will have to rely on lighting and shadows to help define limits of objects. What you will often see is a loss of definition between a foreground and background object with the same color. As shown in Figure 14.3, graphically the difference between the concrete column in the foreground and the concrete wall in the background is hard to discern.

FIGURE 14.3
Can you tell the difference between the column and the wall?

So when using synthetic materials—that is, materials with no patterns—it is important to use variation in the colors to aid in the visual differences. As shown in Figure 14.4, all your concrete object types should have slightly different colors so that they contrast against one another.

FIGURE 14.4 You can use different tones to indicate object types more readily.

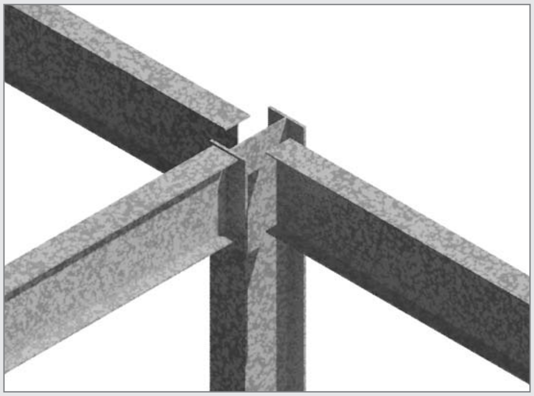
Real World

Most people would prefer to see some sort of pattern on the rendered objects. It could be a concrete with aggregate showing, a masonry wall with grout lines, or steel with some weathering. Anything you can apply to an object that mimics its real-world properties will add a level of realism to your renderings. Take a look at Figure 14.5; this is a real photograph of a concrete surface.

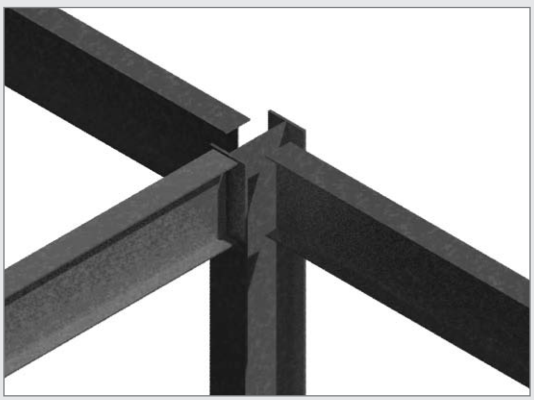
FIGURE 14.5 When applied to objects, a real-world material (image) will make those objects look realistic.

EXERCISE: IT'S ALL MATERIAL

Applying materials to your various objects is fairly easy. In this exercise, you will experience firsthand the steps required. As shown here, this model has a steel material applied but lacks a lot of style. This exercise will determine what material is applied to the steel objects and then change it accordingly.



1. Access MATERIAL.RVT at the book's companion web page (`www.sybex.com/go/masteringrevitstructure2009`). This is a small model with a few beams and a column.
2. On the menu bar choose Settings ➢ Materials to open the Materials dialog box.
3. Scroll down in the Materials panel and select Metal – Steel – ASTM A992. This is the material assigned to steel beam and column families in this model.
4. Once the material is active, select the Render Appearance tab at the top right of the Materials dialog box. A quick scan of the panel will show that the style of the material is based on Aluminum Anodized Dark Bronze. This is the default in Revit Structure 2009 and probably the first thing you should change in your templates.
5. At the top of the panel, click the Replace button Then, using the Render Appearance Library, type **steel** in the Search box at the top. This will reduce the available materials to ones with "steel" in the name.
6. Now locate and pick Stainless Steel Brushed, and then click OK to close. The Materials dialog box now shows Stainless Steel Brushed as the appearance. Next we want to change the shininess of the material.
7. Around the middle of the dialog box is the Finish drop-down list. Click on it and select Satin. Then click OK to close. Now this revised steel material has a more realistic appearance, as shown here:



Once you have materials properly applied to your model objects, as shown in the final illustration of the preceding exercise, yours will be ready at a moment's notice to create renderings. Material configuration and assignment to objects is one of the primary purposes of a template file. You can spend time doing this once and then leverage that time over and over for each following project. The key is to know how your future renderings should be set up. A great example is steel. It can come in many surface styles, from weathering steel (orange rust) to gray primer, and many other colors as well. A good protocol is to create a series of material recipes to recall when need.

You may think that just duplicating Metal – Steel – ASTM A992 over and over for each style would be a good method, but there isn't a good way to change all steel objects from one steel material to another. The quick method is to change the primary material Metal – Steel – ASTM A992 to whatever style you want for most of your objects. Then, if needed, create additional materials for the nontypical steel objects.

Now that your objects are properly materialized, you can begin to create renderings. The following section will explore the ease in which you can obtain great imagery from your Revit Structure models.

Rendering

A first step in understanding how to render is to know *what* you can render. Not every view in Revit Structure can be used for this. Only isometric and camera views can be rendered. But that doesn't mean you can't render "flat" views—there are methods to follow for those as well.

To find out if you have a rendering option on a given view, look at the View Control bar for the view. Along with the scale, detail level, and other controls is an icon to open the Rendering dialog box. As shown in Figure 14.6, the icon image is a teapot. This is in deference to the original teapot model used by many to develop their skills in this artistic arena.

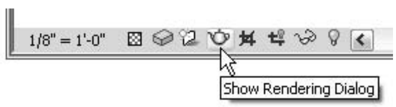


Figure 14.6
Look for the teapot to know if the view can be rendered.

Before you begin to render, prepare your system for the best possible performance. The mental ray engine can use up to four computer processors for rendering computations, so whatever you can do to reduce other active applications will help. Shut down your mail client and anything else not required during the time you would be rendering.

Additionally, in your model you should hide unnecessary objects in the view you will be rendering. These might be objects on the far side of the model but that appear in a wireframe view. If you can see it in the wireframe view, it will be rendered and, if applicable, covered by object faces in front of it. So why bother to render them—just turn them off.

You can also gain some performance by adjusting your detail level from fine to medium or coarse. For example, steel beams have a filleted corner at the web/flange interface. If your point of view is outside the building, this curve will be small and likely not visible. But the mental ray engine will see it and develop a rendered solution for it. Again, increase performance by reducing the detail level. Finally, you can just adjust the area to render by using a crop region or a section box, or by assigning a render region. (A render region will be discussed a little later in this chapter.)

Now that you have properly prepared your to-be-rendered view, you can click the teapot icon or use the menu View ➢ Rendering dialog box. Additionally, you can use the Rendering bar, which is not on by default. If needed, use the right-click menu above the Design bar, as shown in Figure 14.7, to turn on the display of the Rendering bar.

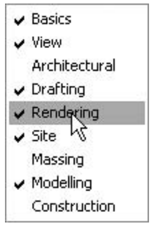


FIGURE 14.7
Use the right-click menu to control the display of Design bar menus.

The Rendering dialog box might appear a bit foreign to you. It contains controls that are unlike anything else in Revit Structure. As shown in Figure 14.8, the dialog box has several zones of adjustments to enable you to create renderings to meet your needs.

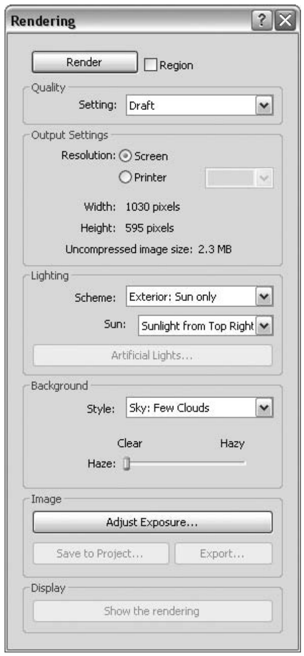


FIGURE 14.8
The Rendering dialog box is used to make all rendered images.

At the very top of the dialog box is the Render button. Perhaps this should have been placed at the bottom, as you click this button only once you've made all your setting changes. Just to the right is a Region check box. Click it, and Revit Structure includes a rectangular region within your active display, with grips on all edges and corners, as shown in Figure 14.9. These grips allow you to adjust the area of the view that the rendering is to be made.

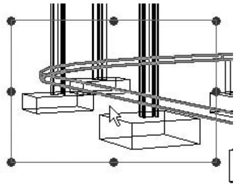


FIGURE 14.9
An example region to render is defined.

Once you have made your adjustments to the region, you simply return to the Rendering dialog box. If you turn off the Region check box and turn it on again, the region will reset to the view default.

Depending on your rendering needs, the quality of the output you require can vary easily using the Quality controls. The following section will demonstrate the types of quality and how to achieve them.

Quality

By default, Revit Structure provides five levels of quality: Draft, Low, Medium, High, and Best. Most users will settle for using only two or three forms, depending on the quality and time to complete they can accept. For purposes of discussion, we ran a series of renderings on three settings. A rendering at Draft (see Figure 14.10) level took 1 minute, 3 seconds, to complete. A Medium level I (see Figure 14.11) rendering took 7 minutes, 49 seconds, to complete. Then, a rendering made at the Best level (see Figure 14.12) took over 2 hours!

FIGURE 14.10
Draft level of quality

FIGURE 14.11
Medium level of quality

FIGURE 14.12
Best level of quality

In all three of these, the resolution was no different (1026×664) and each with a 16 million color palette. The difference is the level of quality of the computations for the rendering itself. The better the quality, the fewer mistakes, known as *artifacts*, that are left in the image. As shown in Figures 14.10, 14.11, and 14.12, each subsequent shot improved pixilation and thereby smoother color blends. The full images are available for your review on the book's accompanying web page; look in the Chapter 14 exercise folder.

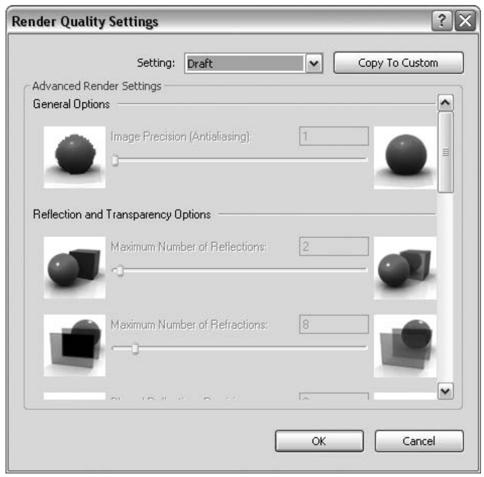
In most cases you should use Draft to define your basic lighting and area of the image. That way, once you're ready, rendering at Medium will produce reasonable results for most digital

uses such as a Microsoft PowerPoint slide. When you need to print an image, then (and only then) you would commit the time for a Best rendering level. Remember that the resolution isn't the issue—it is the quality of the output and how "clean" it looks.

If you review the color images, you will notice that at Draft level the grass is very spotty. But at Medium and Best you can detect little to no difference in the grass. This is because the grass is actually an image, or grass tiled onto the topography object. Also notice the jagged edges on the Draft image that don't exist on the other two.

If none of the default quality settings work for you, or you just want to delve deeper into controlling the quality, all you need to do is click the Edit button in the Setting drop-down list to open the Render Quality Settings dialog box shown in Figure 14.13.

FIGURE 14.13
You can use the Render Quality Settings dialog box to create your rendering quality level.



The Rendering Quality Settings dialog box has several sections available for adjustment. Preset quality levels are included that provide you with a way to tell what is set for each type. For example, you can tell that at Draft level the Image Precision setting is at 1 (jagged edges), but if Medium is selected the Image Precision setting is at 4. So if you just need the speed of Draft but want fewer jagged edges, choose the Custom setting and then set Image Precision to 4. Then click OK to assign those settings and return to the Rendering dialog box.

Here are the various settings for Render Quality:

Image Precision Adjust this value to lower the number of jagged edges in the rendered view. Values range from 1 (very jagged) to 10 (least jagged).

Maximum Number of Reflections Increase this value if objects are not shown within reflections of a rendered view. The range is from 0 (none) to 100 (absolute most available).

Maximum Number of Refractions Adjust this value when objects don't appear through multiple planes of glass. Values range between 0 (opaque) to 100 (transparent).

Blurred Refractions Precision Adjust this value when object edges or surfaces in blurred reflections are spotty. Values range from 1 (spotty) to 11 (smoothest).

Enable Soft Shadows Choose this option to permit shadows to be soft using the Soft Shadow Precision control.

Soft Shadow Precision If available, this option allows shadows to be set from 0 (spotty) to 10 (the smoothest).

Compute Indirect and Sky Illumination Choose this option to permit light from the sky and object-bounced light into your scene.

Indirect Illumination Smoothness Adjust this to provide more detail on objects that are in shadow and lit by indirect lights. Values range from 1 (least detail) to 10 (most detail).

Indirect Illumination Bounces Adjust this value to permit objects in shadow to be lit by indirect lights. It controls how many times a light ray can bounce from object to object. This can lighten objects totally in shadow so that they can be seen. Values range from 1 (single bounce for lights) to 100 (highest number of bounces), but generally more than three bounces is not perceptible.

Daylight Portals for Windows, Doors, and Curtain Walls These controls only apply for daylight within an interior view and provide a means to get light in through the opening.

There are many components to render quality—luckily you can bypass most of them on your way to getting output. In the next section we'll examine output, settings, lighting, and backgrounds.

Output Settings

Once you have specified your quality level, you then determine the output required. The Rendering dialog box offers two options: Screen and Printer.

Screen will assign a resolution based on the visible model view. If you have a maximized viewport, the resolution will be as high as it can be. If you cascade or tile your views, the resolution will change. The resolution is directly related to the visible portion of the screen. For this reason, it can be difficult to hit specific resolution aspect ratios.

The only other option, which frankly isn't much better, is Printer. It allows you to render to a specific dpi (dots per inch). Available options include 75, 150, 300, and 600 dpi. If you need a specific resolution, you will have to render larger than you need and crop with an image-editing application such as Adobe Photoshop. Using an image-editing application to resize the image will lower quality since the image will be pixilated.

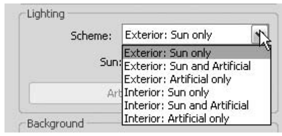
For very large image resolutions, you will need to render in another application altogether, such as Autodesk's 3ds Max. Programs like 3ds Max permit finite control on image resolution and also have better material and lighting controls than Revit Structure.

Lighting

Lighting with Revit Structure is probably the one area where you will not need to adjust much. Structural projects are generally designed using columns rather than solid perimeter walls, so exterior lighting (the sun) typically can be used to generate the light needed for the rendering. Additionally, most structural firms are quite busy with the normal building work and don't have time to delve into light placement. (An exception to this is a parking garage designer who is also responsible for garage light placement.)

As shown in Figure 14.14, Revit Structure provides a number of lighting schemes you can choose from. For a typical exterior daytime rendering, you choose Exterior: Sun Only. Selecting a Sun Only option will disable the Artificial Lights control. Likewise, if you choose any of the Artificial Only options, the Sun control list will be disabled.

FIGURE 14.14
Select from various exterior or interior lighting schemes.

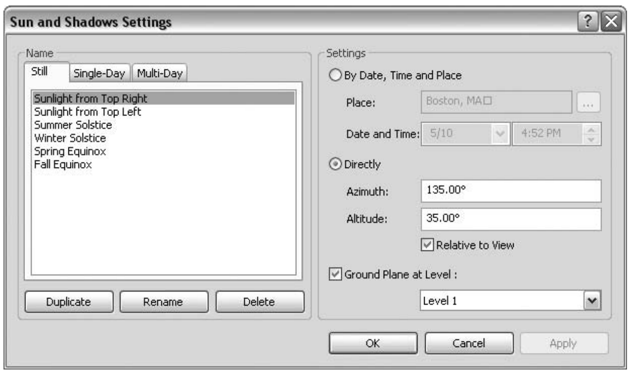


Interior schemes with Sun have the added option of permitting sunlight to enter the scene via windows, doors, and curtain walls. This scheme just allows it; you would still need to use the Render Quality Settings dialog box to include the portal sunlight.

If you choose a Sun-lit scheme, you then have ability to tune the Sun location to whatever your needs are. As shown in Figure 14.15, you not only can point the Sun at the model from a specific direction, but you can also choose to use a yearly position, such as a solstice or equinox. If none of the default Sun locations meet your needs, click the Edit/New button to display the Sun and Shadows Settings dialog box. As shown in Figure 14.16, this dialog box has Name and Settings areas. Depending on what you selected prior to clicking Edit/New, the Still, Single-Day, or Multi-Day tab will be active. If you just need a Still (single) position for the Sun but want it coming from the lower left of your model, you can duplicate the Sunlight from Top Left.

FIGURE 14.15
Sun-provided light can come from many directions, time of year, or relative to any point on Earth.

FIGURE 14.16
Use the Sun and Shadows Settings dialog box to determine the point of origin for the sunlight.



Once you've duplicated the Sun, rename it **`Sunlight from Lower Left`**; then adjust the Azimuth value to 225 (degrees) and deselect the Relative to View check box. Deselecting the Relative to View option will "pin" the location of the Sun regardless of the point of view.

Background

The background you choose in your Revit Structure images is an important issue: you must decide if your goal is realism or clarity, print or digital presentation, or another personal preference. As shown in Figure 14.17, Revit Structure offers up to six styles to choose from: No Clouds, Very Few Clouds, Few Clouds, Cloudy, and Very Cloudy.

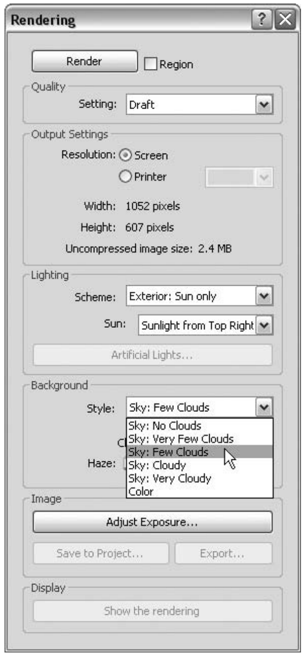


FIGURE 14.17 Revit Structure can give you a cloudy day if you want!

The No Clouds style is simply a fading gradient from a horizon gray to sky light blue. This is generally a great choice and allows a modest level of realism but without the distraction of visible clouds.

The other cloud styles all include increasing cloud volume overlaid on the gradient sky. The last option is Color. When you choose Color, the Haze slider control changes to a color swatch with access to a Define Color dialog box. In the Color dialog box you can set a simple solid color, or define one using Red/Green/Blue or Pantone catalogs.

Your choice should reflect your intended use for the image. Here are some good guidelines:

Print (on paper) Use a solid white background so that the model pops off the page. Color printers are notoriously bad when it comes to fill quality. If you must have a background image, then try to use a gradient with few clouds rather than a solid color.

Digital (on screen) You set this depending on whether the display will be seen in a darkened room or a bright expo hall. For dark conditions, use a solid black background so that the whiteness isn't overbearing to the viewer. In well-lit rooms, choose a white background so that you have good contrast visually.

Obviously, when creating a render the end use could be just about anything. You would be wise to run renders with solid black, white, and gradients. And then for good measure, export the black background image as a PNG so that you save a version with a transparency layer. Later, in an image-editing program, you can place your building in a site photograph by using a masking and transparency layer, or you can even place it into a photograph with you in it (see Figure 14.18). The next section discusses the PNG image format.

Figure 14.18
Get up close and personal with your models!

As you can see, you can take your render to a whole other level of interest.

The last control for backgrounds is a foreground element. The Haze slider control permits you to put a level of fuzziness or fog into your render. This feature is most useful if you are designing a roadway structure, such as a large span bridge, or a major metropolitan high-rise structure.

Dealing with Images

Once you have a rendering made, you then must decide what to do with the fruit of your labor. This section exposes a few options that come into play once you have successfully rendered and are mostly satisfied with the results.

Adjust Exposure

As you begin to render models, take some time to go into the real world and look around. Develop an eye for what lighting looks like, and see how the sun and its contrast affects what

you see. It is contrast that gives your renderings depth. Sure, you can render a model and include every little detail. But that isn't how the real world works. In the real world, some things are clearly visible, while others are hidden in darkened areas. This is where Adjust Exposure comes into play.

As shown in Figure 14.19, there are a multitude of things you can tweak to improve your renderings. In general, you will want to darken a given setting rather than lighten it.

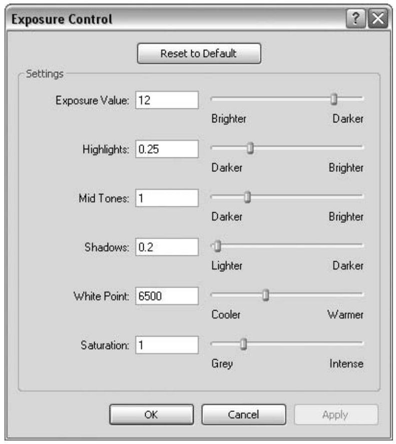


FIGURE 14.19
Adjust the visual contrast within your images by using exposure controls.

Using exposure doesn't require repeated renderings—just one. You simply open the Exposure control before rendering, reset the default values, and then click OK to close. You then render your scene at the quality level you want. Once again, you open the Exposure control and then modify the settings and click Apply. Your rendered image will update accordingly. Once you have the look you want, you can close the dialog box by clicking OK and then save/export as desired.

SAVING TO PROJECT

If you intend to use your renderings in your project sheets, you can click the Save to Project button to place the image in a project view, and then you can drag it onto a sheet view.

This is a great feature—you can update the image simply by resaving to the project and using the same name again. Doing so will overwrite the image, and once your return to the sheet, the revised image will be there.

EXPORT

Once you have your rendering, you probably will want to save it to a file you can use later. Revit Structure provides a number of image file types for you use, but they are not all created equal. Some are best suited for viewing, others for print media, and some for use within other applications.

The choice comes when you click the Export button in the Rendering dialog box. A standard Windows file access dialog box appears that lets you save the image. As you can see in Figure 14.20, the default filename matches the view name being exported. The Save Image dialog box also contains a Files of Type drop-down list.

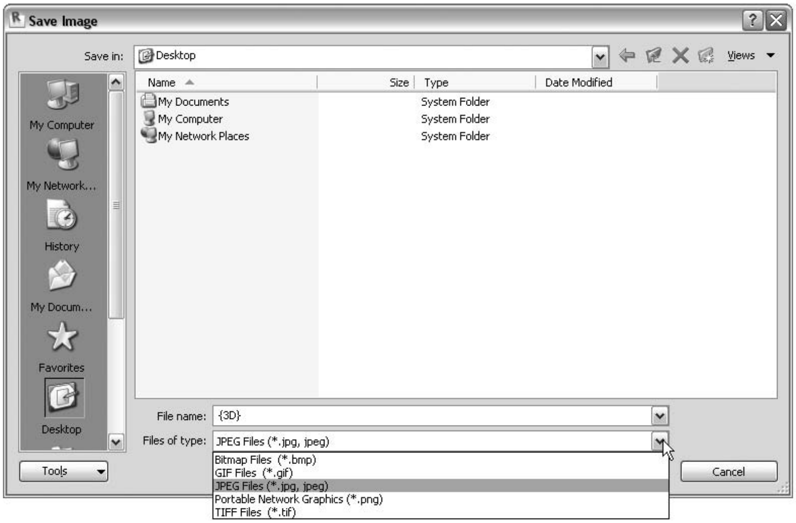


FIGURE 14.20
To save an image, use this Save Image dialog box.

Bitmap files (`*.bmp`) are an industry-standard format developed by Microsoft. The format is well known, and nearly every image processing application can use BMPs. The format does not have a patent, which ensures its widespread use. However, the format tends to be uncompressed and therefore the images will often be very large and usable only for postprocessing applications.

GIF files (`*.gif`) are an image standard developed by CompuServe for use within its network. Originally very popular due to their ability to contain animation, GIFs are now somewhat antiquated due to their color depth limitation of 256. GIF does still have its uses, since GIF files can contain transparent layers that can be used in postprocessing to aid in masking between photographic and render image overlays.

JPEG files (`*.jpg`, `*.jpeg`) are today's standard for creation of user-ready images. The format has multiple compression levels and color depth controls, and is generally well suited for Internet web presentation and standard-sized print uses. Due to its aliasing (pixel stepping), the JPEG is not suited for animation compiling. However, JPEG images can often be reduced in size while retaining a quality appearance.

The Portable Network Graphics (`*.png`) format, the replacement for GIF, offers increased color depth but no animation options. It also does not have a patent and so has been refined over the years into a great option. The color protocol is only RGB, but since it contains a transparency layer, the PNG can be used for masking easily.

TIFF files (`*.tif`) are the workhorse of the professional medias. The file format tends to be very large and the images can be lossless if desired. These images types are not well suited for Internet use; since they are formatted in CMYK color, they are heavily used where color-offset printing is done. A TIFF is often the best choice for postcompiling animations based on sequential images.

So, which do you use? If you are creating a single image for the Web or small 4 × 6 prints, use a JPEG. If you are creating an animation and intend to compile it postrender, use TIFFs for the images. If you are creating a tiny icon, use a BMP. If you desire an image you can print at 24″×36″, again, use a TIFF for the best quality, or if size is an issue, use JPEG. Finally, if you need to use the image layered with something else in postrender, then choose PNG.

Controlling Display

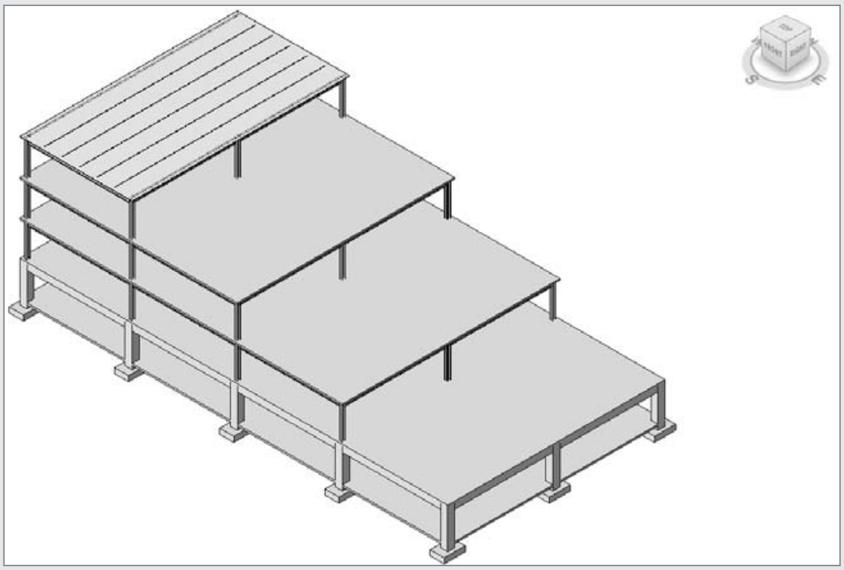
The final option to examine in the Rendering dialog box is the simplest. The Display portion contains a single button control. Prior to rendering an image, this button is disabled. Once you have successfully rendered the view, this button reads Show the Model. And it does just that. When you click it, Revit Structure will clear the rendering from the Display area and replace it with the previous view of the model. Want to see the rendering again? Simple—click the Show the Rendering button and it comes back. We recommend that you save the model before switching back and forth between showing the model and showing the rendering.

You can close the Rendering dialog box, switch to other views, and as long as the rendered view stays open, you can recall the last rendered view anytime. Once the model or view is closed, the rendered view will be discarded. Use Save to Project or Export if you need the rendered view later.

As you now know, there is a lot of variation and control over the final product of your renderings. Next up is an exercise where you will get the chance to experience the process firsthand.

Exercise: Rendering Your Model

Now that you have a complete picture of the rendering process, it is finally time to render! The model you will render is sized for speed and ease of use; yours will be much more interesting! There isn't a whole lot to successful renderings; just remember that beauty is in the eye of the beholder (see the following illustration).



1. Open `RENDER.RVT` from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`.

2. Using the Project Browser, activate the 3D View ISOMETRIC.
3. At the bottom of the ISOMETRIC view, click the Show Rendering Dialog button (the teapot).
4. In the Rendering dialog box, change the Quality setting to Medium.
5. Click the Render button at the top to create the rendering.

Depending on the speed of your computer system, after a minute or so the ISOMETRIC view window will render. Odds are it won't look all that great. It will be washed out since no settings have been changed—yet.

1. With the Rendering dialog box still open, click Adjust Exposure.
2. Click the Reset to Default button at the top of the dialog box.
3. Change the Shadows value to 3 and then click Apply.
4. Click OK to close.

Instantly the image will darken, gain contrast in the shadows, and become more acceptable. Next we will adjust the materials.

1. Choose Settings ➢ Materials.
2. In the Materials list panel, locate and select Concrete – Cast-in-Place Concrete.
3. Select the Render Appearance tab and then click the Replace button.
4. Using the browser window, find and select Concrete. Then click OK to apply and close.
5. Scroll the Materials list panel to find and click the Metal – Steel – ASTM 992 material. Click the Replace button.
6. In the Search box, type **`Paint Dark Red Matte`** and then click OK to accept and close. The Preview will change to a wall corner.
7. Click OK to close the Materials dialog box.

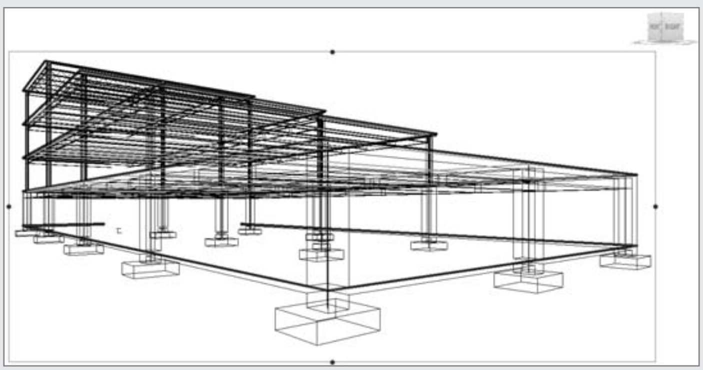
You now have changed the material for concrete to a less dense pattern. And now steel is a red primer color. Time to adjust the sun and render again!

1. The Rendering dialog box should still be open; if not, click the teapot again.
2. In the Lighting area, click on the Sun list and choose Edit/New.
3. In the Sun and Shadows Settings dialog box, click Duplicate with the Sunlight from Top Right option selected.
4. Name the Sun location **`Sunlight from Lower Left`** and click OK.
5. With Sunlight from Lower Left selected, change the Azimuth value to 225 and turn off the Relative to View check box.
6. Click OK to apply and close.
7. In the Rendering dialog box, click the Background Style list and choose Color. You then get a color button; click it and assign a black color. Click OK to close.
8. Change the Quality setting to High and then click Render. Now get some coffee or check out `www.augi.com`. This will take a few minutes.

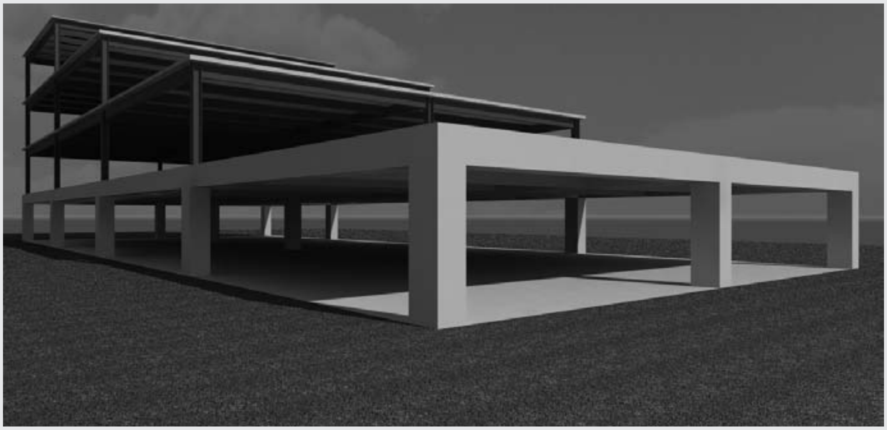
You have a good-looking rendering with a nice level of contrast and obvious steel elements, and it's ready for a video presentation, as shown here.

Revit Structure offers two view types to render by. One is the 3D view you just witnessed. The other is by using a camera. Next, you will do that and then render from the ground point of view and add some clouds.

1. Using the Project Browser, activate the Level 1 plan view.
2. Using the menu bar, choose View ➢ New ➢ Camera.
3. Note that the Options bar now contains controls for camera setting and placement. The default values are fine; start with a point on the plan view to the lower right of the building.
4. After you select a camera point, hold down the left mouse button and drag the new target point to the middle of the building plan.
5. Once you set a target point, a new camera view will be generated and displayed for you, as shown here. Then you can use the grip controls to fine-tune your view. Adjust it so that the entire building, including footings, is shown within the crop box.



6. With the new Camera view active, choose View ➢ Visibility/Graphics. Then in the Visibility/Graphics Overrides dialog box, select the Show Categories from All Disciplines check box. Then scroll down the Model Categories list for Topography and select it. Click OK to apply and close.
7. Now open the Rendering dialog box via the teapot.
8. Change the Background Style setting to Sky: Very Cloudy.
9. Open the Adjust Exposure control, reset the values, and change the Shadows value to 3. Click OK to apply and close.
10. Change the Quality setting to High. When ready (when you have the time!) click the Render button and once again take a break while it generates. The following graphic shows the final rendering.



Now that you have finally rendered a view, you can sit back and enjoy the fruits of your labor, right? If you are lucky, maybe! Most likely you will have more modeling to do along with rendering new images. A great technique is to create some cameras within in your model and use them repeatedly to always have a latest version fully rendered, ready for review.

But single-frame renderings are not the only thing you can do with this technology. As the following sections will show, there are some nontraditional uses for rendering your models.

Sun Studies

Also included in Revit Structure is a means of creating single- and multiple-day Sun studies. In the Sun and Shadows Settings dialog box is a tab area for Single-Day and another for Multi-Day. In general these configuration options are not a concern for the structural designer; their inclusion in the software is due to the fact that Sun studies are a part of the core Revit program.

The steps to create a study are simple:

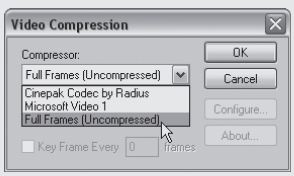
1. Activate the 3D perspective view.
2. Using the View Control bar, open the Model Graphics Style pop-up list and assign Shading or Shading with Edges.

3. Choose View ➢ Advanced Model Graphics, then in the Advanced Model Graphics Settings dialog box, click the ellipsis button.
4. In the Sun and Shadows Settings dialog box, choose either the Single-Day or Multi-Day tab. Click the Duplicate button and name the new location the same as your model location.
5. With the new location highlighted, click the ellipsis button in the Settings area.
6. Using the Manage Place and Locations dialog box, locate your city or assign custom Latitude/Longitude values. If required, turn on the Adjust Daylight Savings check box. Click OK to close.
7. Click OK to close the Sun and Shadows Settings dialog box.
8. Click OK to close the Advanced Model Graphics Settings dialog box.
9. Using the Shadows On/Off control (to the right of Model Graphics Style), turn Shadows On.
10. Choose File ➢ Export ➢ Animated Solar Study.
11. Using the Length/Format dialog box, adjust for desired frame length as well as image visual style and size.
12. Assign a name and location for the exported file(s). If you choose the AVI format, the Video Compression dialog box will allow you to assign compression and codec format.

That's it—you've created a Sun study. Frankly, not many will use this feature of Revit Structure, but it is good to know anyway. Next we will move into how to create walkthroughs of your models.

Video File Exports

Revit Structure AVI files typically are very large. To combat this you can use either the Cinepak Codec by Radius or Microsoft Video 1 compression. Although most animation professionals compile videos after rendering the animation to a series of sequential images, exporting your frames to individual TIF or another uncompressed image file type (see the following graphic) and then compiling into an animation via something like Adobe Premier will not only give you better quality and access to many more video file types but also permit effects and other after-render changes such as resizing, watermarks, and soundtrack mixing.

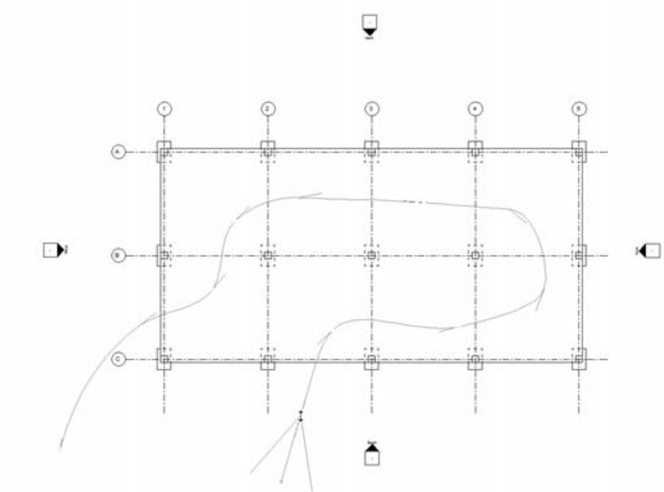


The Walkthrough Command

An underused feature of Revit Structure is the Walkthrough command. Use this command to create a path that a camera will follow; the command then exports single frames or an animation of what the camera sees as it moves along. The basic technique is not hard to apply; the key is to remember that the tool is there to be used.

1. Activate a plan view and then choose View ➤ New ➤ Walkthrough.
2. The Options bar then populates with Walkthrough settings. Generally a default of 5′-6″ as a head height is appropriate. You can also change the level you want to use as the basis for the walkthrough.
3. Click points on the plan; each successive point will move the camera and begin to define a spline-like pathway for the camera, as you can see in Figure 14.21.

FIGURE 14.21
A walkthrough pathway on the plan



4. Once you complete the path, click the Finish button on the Options bar.
5. Once the path is created, it will appear highlighted on the plan. If you deselect it, it will vanish from the plan view. Just as with a camera, you must select a walkthrough for it to be visible on the plan. And just like any other view, it will be stored in the Project Browser.
6. Using the Project Browser, find and select the Walkthough view. Then click Element Properties to change its settings.
7. In the Element Properties dialog box, change Model Graphics Style to Shaded with Edges.
8. Turn off the Far Clip Active option, thus allowing for full depth views.
9. Scroll further down and click the Walkthrough Frames button.
10. In the Walkthrough Frames dialog box, shown in Figure 14.22, change the total frame count to 200. Click OK to close.

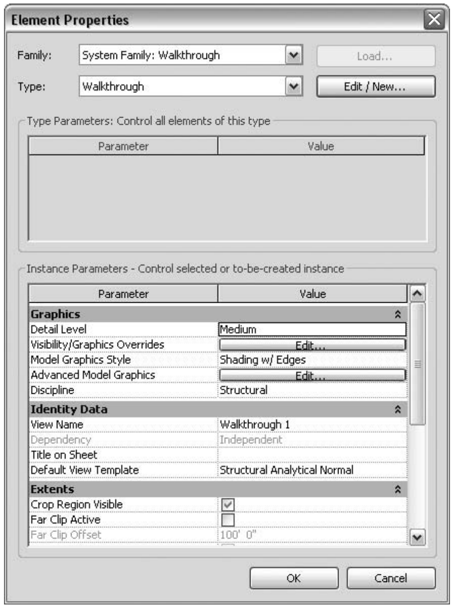


FIGURE 14.22 Adjust your walkthrough as required.

Next we'll create the walkthrough animation:

1. Using the Project Browser, locate your walkthrough and activate it. Depending on your view, you might not see anything at first.
2. Choose File ➢ Export ➢ Walkthrough.
3. The Length/Format dialog box will appear, which allows secondary changes to your animation timing and style. Just click OK to accept the defaults.
4. The Export Walkthrough dialog appears, where you can specify if you want to create sequential images for later compiling or accept the default AVI format. For most walkthroughs, a Revit Structure–generated AVI will be fine.

The purpose of a walkthrough is to study the model for coordination needs; it typically isn't for high-quality presentation, as shown in Figure 14.23.

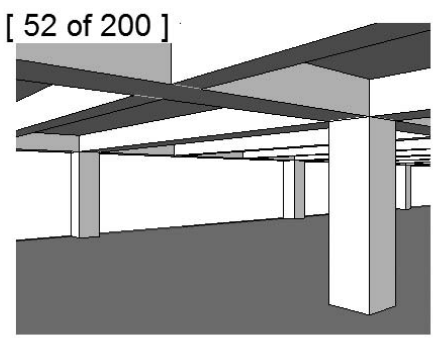


FIGURE 14.23 A single frame of the model shows the reduced level of quality along with index text to show you where along the walkthrough you are.

As you now know, you can get just about any view of your model out into some sort of rendered view. You can create images from basic quality level and style all the way up to high resolution and full shadows. Once you have the tools on your computer belt, you have to apply them every chance you get.

For example, try taking advantage of how you break up your models. If you are doing an addition to an existing building, create another set of materials and object types for existing structure in a flat gray, along with the new structure. Then when you render, you can present a clear difference between the old and new structure.

Go to the Paint Store First

Knowing the materials that look good and meet your needs can save you lots of time. We spent tons of time assigning materials we *thought* met the client's needs. We then spent tons of time rendering stills for a 10-second animation. A single second of animation is typically 30 frames or stills. So when it took 5 minutes per rendering, it actually took 5×30×10, or 1,500 frames—which translated to 25 hours of rendering! If you don't have the proper materials, you can lose a whole day of computer time, not to mention experience a lot of frustration.

Exporting for Outside Rendering

Revit Structure comes with mental ray, by far the best rendering solution provided within the software to date. However, as you get better at developing rendering scenes, you might want a bit more control over the finished product. Revit Structure is natively a modeler, a BIM solution, and a great documentation tool; it isn't a superior program for rendering images.

A number of awesome solutions are available for rendering images of your models, and many of those programs are created by Autodesk. Most notably is 3ds Max, a longtime tool of professionals worldwide. With the introduction of the 2009 class of programs, a new version also became available: 3ds Max Design. This application shares the same core platform as 3ds Max 2009. But in the future it will be tweaked for the Revit user and perhaps solve some nagging deficiencies.

RVT Is Not Supported

The native file format for Revit Structure is not supported in any outside application. Due to this limitation, you must export the appropriate 3D model view to another file and then use that file to import into the rendering program.

At this time the best methods for getting model data into an outside rendering application is via DWG and the newest format, FBX. Each provides the basic goal exporting model geometry into the rendering program, but each delivers different benefits, as you'll see in a moment.

AutoCAD DWG

The AutoCAD Drawing format DWG is probably the most-used format. It carries with it all the 3D geometry and, depending on the rendering application, can read camera and light elements. When exporting, you have a few options, such as specifying the type of objects you want in the newly created DWG file. Access the Export Options dialog box, shown in Figure 14.24, by clicking the Options button in the Export CAD Formats dialog box.

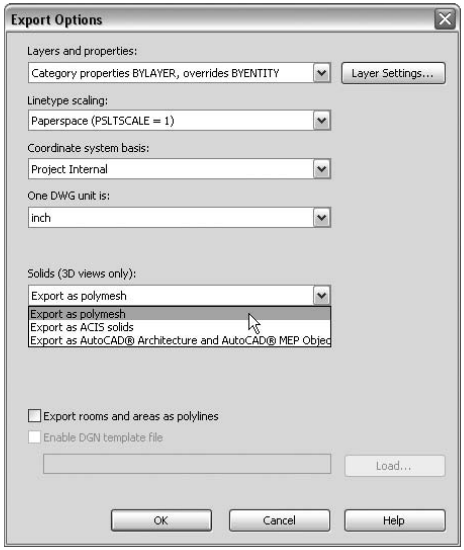


FIGURE 14.24
Limited yet functional export formats

If you choose Export as Polymesh from the Solids drop-down list, most of the model objects will be saved as blocks in the DWG. Each block (W-Wide Flange-Column – W10X49_1-196959-3D View 1) is named rather intelligently, with the family name, type, unique ID, and the name of the view tacked on the end for good measure. Note that one of the odd things about this format is the block. Not all elements are bundled into blocks. Regardless, all model objects below the block level are polymesh. If you then exploded the polymesh objects, you would be left with elements. This format will also provide centerline geometry for all structural objects. This can be helpful if you need to model something natively within the rendering application since you will have the wireframe to build upon. Export as Polymesh is best suited for going right into the rendering application. If you will not be editing the DWG, choose Export as Polymesh.

If you choose Export as ACIS Solids, all model objects except topography will be exported as a mix of blocks and ACIS solids. If you explode the blocks, the resulting object is a solid. As with the polymesh option, you will also get a centerline for all structural objects. This ACIS object type is well suited for editing within AutoCAD since you can use many solid editing tools to further refine your model. If you need to work on the model prior to rendering, or even if you're just going into AutoCAD, then use Export as ACIS Solids.

If you choose Export as AutoCAD Architecture, Revit Structure will try to make all model objects ADT (Architectural Desktop) type objects. They will contain object types that ADT/AutoCAD Architecture can understand and manipulate. Most of these objects can be exploded

further. It may take a few explode executions to get the AutoCAD Architecture objects into 3D face objects. So if the exported model is intended to be used with Architectural Desktop or AutoCAD Architecture, then use Export as AutoCAD Architecture. Also, you will need to keep an eye on any radial geometry; the function still has difficulties manipulating this data.

Why all this effort? Because choosing the proper method will help you when you begin your rendering work. Let's assume you will be going from Revit Structure straight into 3ds Max Design. The best choice is polymesh. It will be smallest file type and is already a mesh.

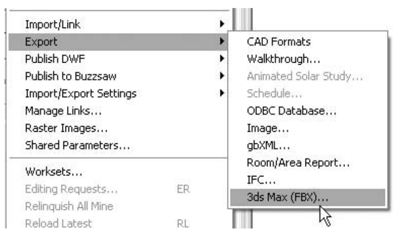
When you import your file into 3ds Max Design, you will be asked how to derive the imported model. By Layer is the best method only if you spent a lot of time organizing your model objects in the DWG file. However, if you are trying to be as time efficient as possible, then use the Entity, Blocks as Node Hierarchy method. The reason By Layer isn't a wise choice normally is because many objects within Revit Structure belong to the same class and therefore the same layer. For example, steel beams and concrete beams are both structural framing and would be on the same layer. But if you derive by entity instead, then you will end up with type name access once inside 3ds Max Design. The type name will be easy to locate and select, and you can then apply required materials.

3ds Max (FBX)

FBX is another player in the import world. This somewhat new format by Autodesk is a first stab at linking the rendering efforts in Revit Structure and AutoCAD into 3ds Max Design. Not only will it bring in the model objects, but it will also import cameras, lights, and materials. As you know, you can choose one of two default rendering engines in 3ds Max. One is mental ray (the same as Revit Structure); the other is V-Ray. You *must* set the default rendering engine in Max to mental ray, or the imported FBX file will not import correctly.

As shown in Figure 14.25, you can find this export option under File ➤ Export ➤ 3ds Max (FBX). You are prompted for the name and location of the newly created FBX file.

Figure 14.25
Export as 3ds Max (FBX)



The only case where using FBX may be worth the trouble is if you are trying to match point-of-view of renders done in Revit Structure. Generally, the materials, lighting, and even cameras are better in 3ds Max Design. So the ability to bring them in along with the model is of little benefit.

Most professional rendering work is done not in Revit Structure but in some other application. The following section will discuss what is involved in rendering beyond Revit Structure.

Outside Rendering

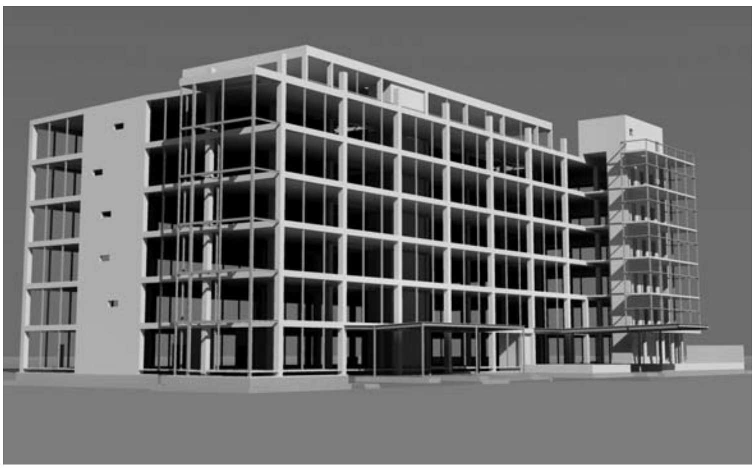
As mentioned earlier, rendering in Revit Structure is a great final touch for your modeling effort. You get a lot of bang for little effort. But you might want a little more from your renderings that Revit Structure can provide.

For example, animations can be made with Revit Structure, but you are limited to planer camera movements along a path. You can also render clouds but are limited to the cloud generation in the software. With outside rendering engines, you gain control on material placement, tiling, transparency, and many other settings that you don't have in Revit Structure. You can also render with radiosity in Revit Structure but only with mental ray; there are many high-quality renderers out there. Object and sub-object animation are not possible with Revit Structure; objects themselves cannot move in an animation. So if you wanted to do a construction sequence animation, you would have to use an outside rendering solution, period.

Covering in detail the methods and applications for these is beyond the scope of this book, but here are few comparisons to consider. In Figure 14.26 is a project rendered a high level with Revit Structure and mental ray.

Figure 14.26
This is a reasonable-quality render made with Revit Structure 2009.

Figure 14.27
The model was exported as a DWG, imported into 3ds Max Design, and rendered with minimal material changes.



Most would agree that there is a markedly improved visual quality in Figure 14.27. For some it would be hard to quantify, as beauty is in the eye of the beholder. But the combination of greater contrasts, smoother materials, more realistic lighting, and translucent ground surfaces provide an additional touch that may warrant investment in non–Revit Structure applications.

The Bottom Line

Determine what and when to model. Once you get going in Revit Structure, the ease of creating models is both a blessing and a curse. If you model too little, you don't achieve the desired result. If you model too much, then you will have so much more than you need, your renderings will take an excessive amount of time.

Master It Before modeling, develop a scope of what and when to model. Conduct team meetings with all project modelers so that everyone involved has rendering in mind as they do their work. Limit the complexity of your renderings by using appropriate detail levels.

Assign materials to your model. Actually rendering in Revit Structure isn't hard—having something render-worthy is the hard part. Materials make or break your renderings. You can make your model look real or *like* a real model.

Master It As you develop your families, assign materials so that you can render on demand later. Using the Materials dialog box, create materials for steel and concrete for when they are viewed at a distance. Adjust materials for rendering even if you won't be rendering now. This will reduce the time needed to prepare for when you are asked to produce images. For real photographic needs, use materials that have few repeating patterns so that no matter the point of view the materials you use will still maintain a level of smoothness.

Define the quality and style of your renderings. When you begin to render your model, you can be overwhelmed with all the settings at your disposal. You can define where the Sun is, what time of day it is, what resolution to create, and how detailed your images should be.

Master It Take a look around in the real world. Get a sense of what structures look like when they are under construction. Things are often dark; you don't always have to light everything up. When you create renderings, save time and only create high-quality at the very end. Use the Rendering system with Autodesk mental ray to define a Sun, adjust exposure, control shadows, and create renderings. Then save your rendering to any number of image types.

Export your models for other uses. Exporting your model for outside use is a typical activity of the true professional. You don't use one kind of writing implement, so you should not use only one rendering application.

Master It Once you have a 3D view active, you can export it to a DWG or FBX file to use in an outside application. Use the FBX format if you have Revit Structure cameras you want to export as well. Use the DWG format with polymesh for direct import into 3ds Max Design. But if you have very large models, you might want to use ACIS solids since that allows 3ds Max to control the meshing directly.

Chapter 15

Revit Structure Analysis

Revit Structure is fully capable of providing all of the information required for analysis. Revit Structure itself will not, however, perform the actual calculations. Within Revit Structure you create point loads, line loads, and area loads. These loads are defined by load cases as defined within Revit Structure. The load cases can then be grouped into load combinations for lateral and gravity cases. These loads are diagrammatically placed into the Revit Structure model using the default analytical views or new views you make during the modeling process.

The analysis model coexists with the physical model but is not necessarily identical. For many reasons the analytical model can be simplified or altered by the structural engineer for analysis needs. Revit Structure analysis is most useful for preliminary analysis of your structure.

Once the loads are in place, you export the model for import into your analysis software. After the calculations have been performed, you then import the analysis data back into Revit Structure. Revit Structure will resize the members according to the incoming data. It is important to note that during this process there is no physical movement of the model itself. This procedure is just a flow of data. You do not have to worry about finding an insertion point or making sure the model is lined up.

In this chapter, you will learn to:

- Configure Revit Structure structural settings and create loads for your project
- Place analytical load patterns onto your model
- Import and export your virtual model from Revit Structure to structural analysis software

Creating Loads

To get started with creating loads, you will step through the process necessary to configure and add load patterns (see Figure 15.1) to your project. First you will need to adjust the structural settings that pertain to the analytical model and configure them for your particular project. Then you will need to set symbolic representations. Once that is completed you will move on to creating the different types of load patterns in your model. This discussion will center on load cases and load natures. Finally you will study how load combinations are used, as well as combination types and states.

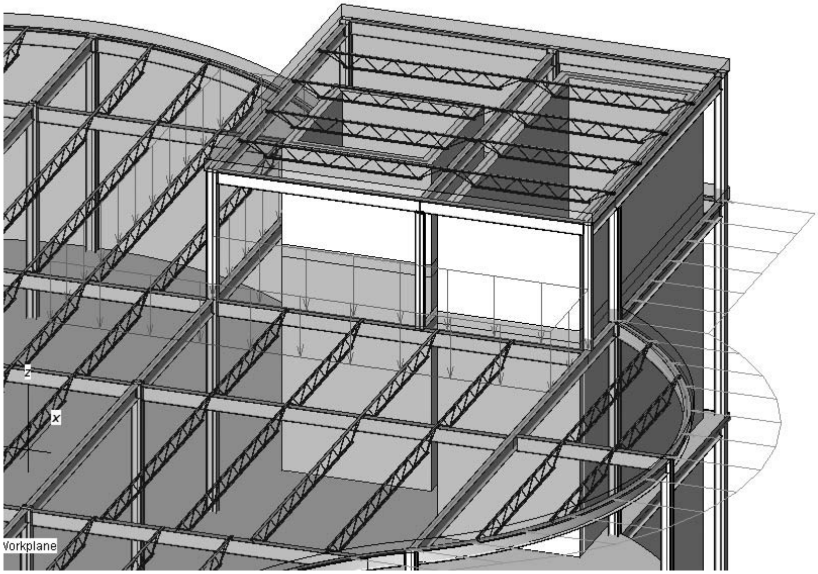


FIGURE 15.1
Various load patterns on a structure

Structural Settings

The first step in preparing the analytical model is to review the structural settings (see Figure 15.2) contained within Revit Structure that pertain to the analytical model. For the procedures described in this section, please use the file `Datasest_1501.rvt` to follow along and to practice the techniques. Otherwise we would have to create all the model elements first, and this is beside the point in our current discussion. To download the files please go to the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009` if you have not done so already. So please go ahead and open that dataset now.

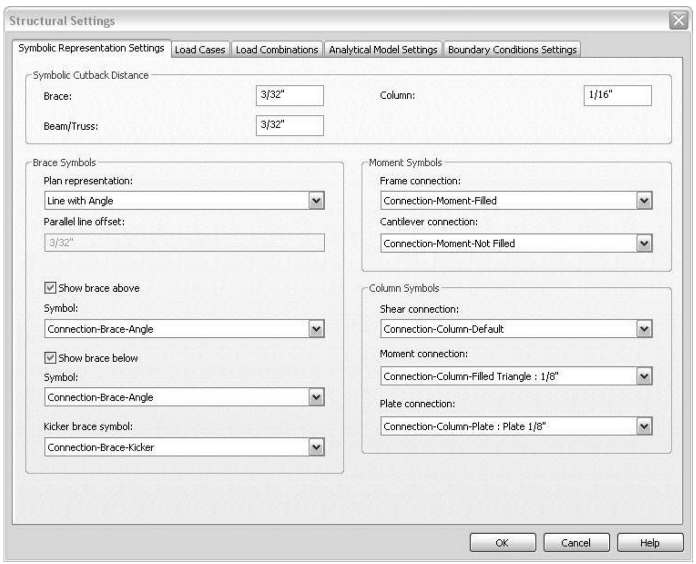


FIGURE 15.2
Structural Settings dialog box

Symbolic Representation

To access the Structural Settings dialog box, click Settings ➤ Structural Settings on the menu bar. Within the Structural Settings dialog box, there are several tabs across the top. We will cover all of these. The first tab is Symbolic Representation Settings. This tab deals mainly with the graphical model and the more common defaults.

Symbolic Cutback Distance represents the gap that Revit Structure will leave between stick symbols. This setting controls the representation of a beam in coarse detail level in order to display a gap between intersecting members, whether it is a column or another framing member (see Figure 15.3). This cutback distance is symbolic or graphic and has no bearing on the real 3D extents of a beam. For instance, in a plan view beams do not extend into columns; there is a gap between them. The same connection in a 3D view will show the beams ½″ back from the column, more consistent with the actual construction techniques (see Figure 15.4).

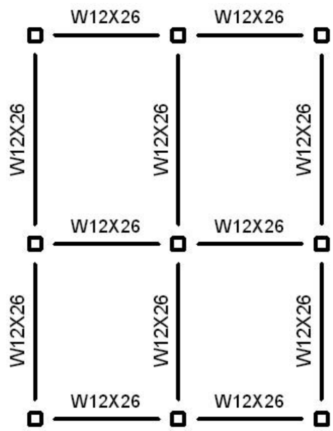


Figure 15.3
Stick symbols show the cutback in coarse detail level

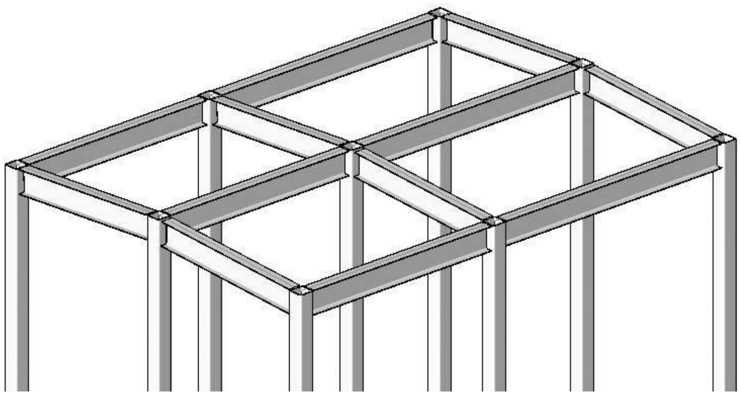


Figure 15.4
Medium and fine detail levels display no cutback.

Load Cases

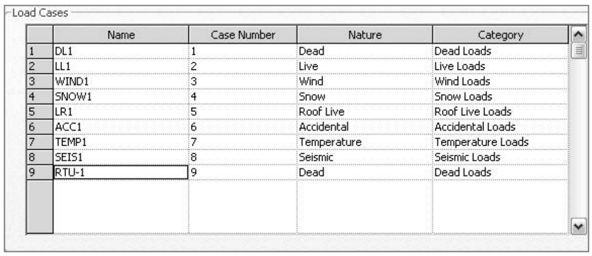
The Load Cases tab (see Figure 15.5) allows you to view and modify existing Revit Structure load cases as well as create new ones. Before you place loads in the model, it is good practice to establish some additional load cases (if needed) here in this dialog box. To create a new load case, follow this procedure:

1. Go back to the open `Datasest_1501.rvt` file.
2. Choose Settings ➢ Structural Settings ➢ Load Cases.
3. In the Load Cases tab, you will see two categories:
   - The top category is Load Cases.
   - The bottom category is Load Natures.

In this example you need a specific load case for an extra-large mechanical rooftop unit:

4. Click the Add button to the right of the Load Cases field.
5. Rename the new load case to **RTU-1**. Its Nature is Dead, and the Category is Dead Loads.

Figure 15.5
The Load Cases dialog box


Load Cases

	Name	Case Number	Nature	Category
1	DL1	1	Dead	Dead Loads
2	LL1	2	Live	Live Loads
3	WIND1	3	Wind	Wind Loads
4	SNOW1	4	Snow	Snow Loads
5	LR1	5	Roof Live	Roof Live Loads
6	ACC1	6	Accidental	Accidental Loads
7	TEMP1	7	Temperature	Temperature Loads
8	SEIS1	8	Seismic	Seismic Loads
9	RTU-1	9	Dead	Dead Loads

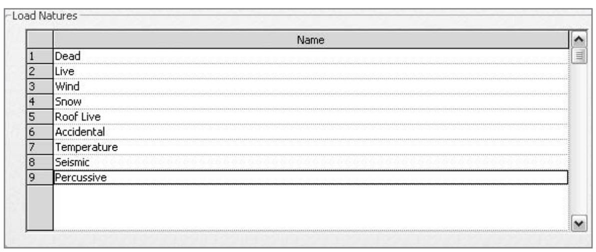
Now you can use this new load case as an area load and place it into the model.

Load Natures

The load natures are added to a load case. The default load natures are the same as you see in most analysis applications. When you create a new load case and a new load nature in Revit Structure, they will be imported as loads into the analysis software you are using.

Normally the existing load natures (see Figure 15.6) will suit your needs for the load cases. If not, you can simply click the Add button and add a new load nature. Simply rename the load nature as required.

Figure 15.6
You can add other load natures to these default load natures.


Load Natures

	Name
1	Dead
2	Live
3	Wind
4	Snow
5	Roof Live
6	Accidental
7	Temperature
8	Seismic
9	Percussive

LOAD COMBINATIONS

In Revit Structure, you can combine load cases. This is a good thing because other analytical software programs in the past have been restricted to gravity loads only. If you wanted a 3D analysis (gravity combined with a lateral load), you had to do them separately. Revit Structure allows you to do gravity, lateral, or a combination of the two. Once you build the load combination, you can then apply it to the model.

1. In Revit Structure, choose Settings ➢ Structural Settings,➢ Load Combinations.
2. Click the Add button to the right of the Load Combination field.
3. Rename the new load case **Wind and Gravity** (see Figure 15.7).

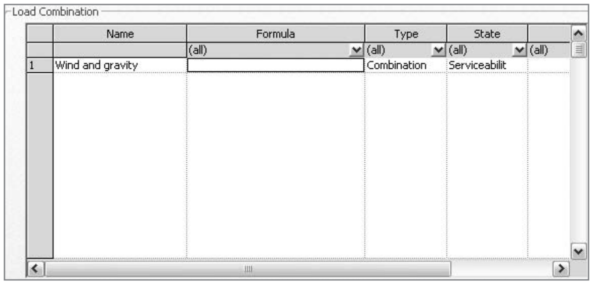


FIGURE 15.7
Creating a load combination

The Formula column contains a drop-down list of the load cases present in the model (see Figure 15.8). Of course this does not help us because they are single cases.

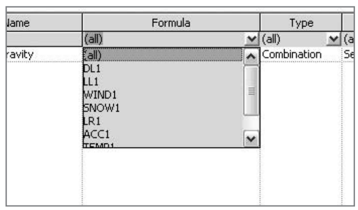


FIGURE 15.8
The Formula drop-down menu

Now you will create a new formula that will combine multiple load cases.

EDIT SELECTED FORMULA

In the bottom-right corner of the Load Cases dialog box, you will see the Edit Selected Formula field (see Figure 15.9). This field will actually combine the load cases. You could consider this as a package of load cases that is inserted into a load combination.

1. Highlight the Wind and Gravity load combination you just created.
2. Click the Add button to start adding a formula.
3. Leave the Factor field set to 1.00000, and set the load to **DL1** (a dead load).

4. Click Add again, and change the factor to **0.750000**.
5. Change the case or combination to **SNOW1**.
6. Click Add again, and select WIND1 as the case or combination.
7. Set the Factor to **0.250000** (see Figure 15.10).

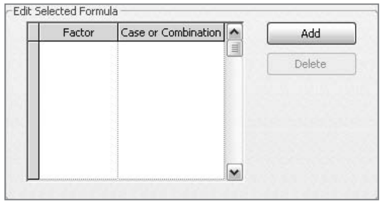


FIGURE 15.9 Use the Edit Selected Formula field to combine load cases.

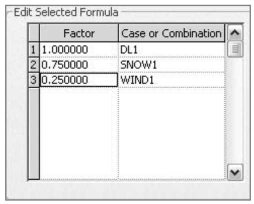


FIGURE 15.10 Adding a formula for analysis

Look up at the Load Combination Wind and Gravity field. Notice that the formula is now present. This formula is an integral part of that particular load case.

LOAD COMBINATION USAGE

This category determines what the load combination will be used for. It is not a formulaic expression; rather it is for a descriptive field. You should keep this usage naming convention consistent with your analysis software wherever possible. This will aid the interoperability as you pass loads to the analysis application and back again.

1. In the lower-right corner of the dialog box, click the Add button in the Load Combination Usage area.
2. Rename it **Large RTUs**.
3. Click the Set check box. This will dedicate the usage to the current load combination (see Figure 15.11).

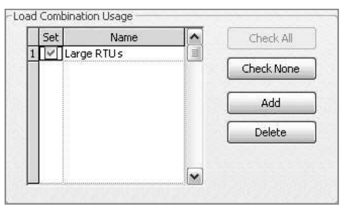


FIGURE 15.11 Dedicating the load combination usage

Load Combination Types

There are two different types of load combination: Combination and Envelope. Setting the load combination type to Combination provides information for a single combination. Envelope gives maximum and minimum results on a group of load combinations. Set the Load Combination type for Wind and Gravity to Combination.

Load Combination States

Revit Structure offers two different states for a load combination: Serviceability and Ultimate. A Serviceability state will categorize the load based on an expected force, such as wind, gravity, natural loads such as snow, and even deflection. An Ultimate state tests the load against unexpected forces and overall stability of the structure when pushed to an ultimate state.

1. Set the Load Combination state for Wind and Gravity to Serviceability.
2. Click OK to accept the changes and exit the Structural Settings dialog box, but leave your model file open.

Now that you have studied the settings for load cases and combinations, you will next learn about the analytical model settings.

Analytical Model Settings

The Analytical Model Settings tab on the Structural Settings dialog box sets the default warnings and is a means to check the structural stability of your model as you work. By default, many of the options are checked for a good reason. For example, if you place a structural column with no bearing footing or pier, a setting on this tab will alert you to the structural deficiency. As you are starting the model, it will, by the nature of the design process, be annoying. Having a warning appear every time you want to place a structural item into the model would become quite daunting and somewhat time consuming.

So in this section you will learn how to configure the automatic checks for your model and to set the correct tolerances. You will see how the analytical model is distinguished from the physical model and how analytical views are created in your project. Then you examine particular model elements such as columns, floors, and walls to see how the analytical components for those objects work. Finally you will learn how to set up boundary conditions within your analytical model.

Automatic Checks

The top of the dialog box allows you to set the automatic checks. The following steps outline the recommended settings for automatic checks:

1. To start, open a brand new Revit Structure project.
2. Choose Settings ➢ Structural Settings ➢ Analytical Model Settings (see Figure 15.12).

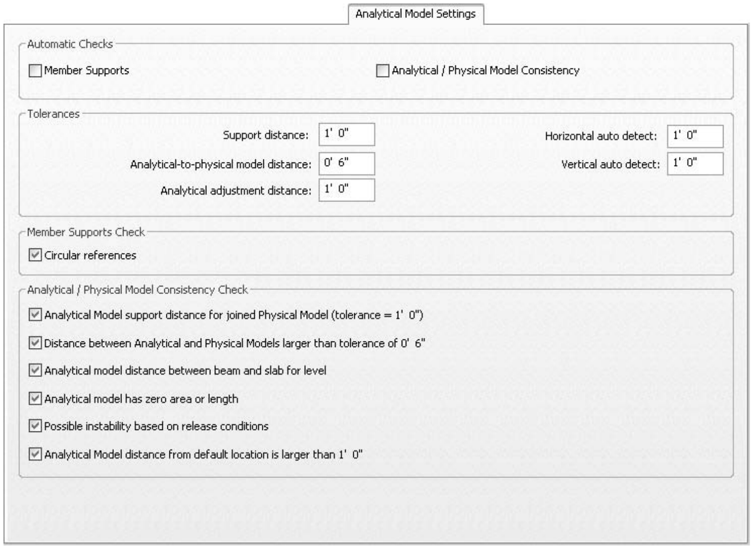


FIGURE 15.12
Adjusting the analytical model settings

There are two items on which you can keep a running check. These items are Member Supports and Analytical/Physical Model Consistency.

3. Check both Member Supports and Analytical/Physical Model Consistency.
4. Click OK on the Structural Settings dialog box to go back to the model.
5. You will be asked if you want to perform an Analytical Model Check at this time. Click No. The model is still blank.
6. On the Basics tab of the Design bar, click the Structural Column button.
7. Place a column in the model. A warning will appear, as illustrated in Figure 15.13.

FIGURE 15.13
A warning appears for the unsupported column.

If you are deliberately looking for unsupported members, Revit Structure has the power and the ability to perform this function. Follow these steps:

1. Select the column.
2. On the Options bar, you will see an exclamation marker. Select it. The same warning is displayed in the Messages dialog box (see Figure 15.14).
3. Click the Close button.
4. Choose Settings ➢ Structural Settings ➢ Analytical Model Settings.

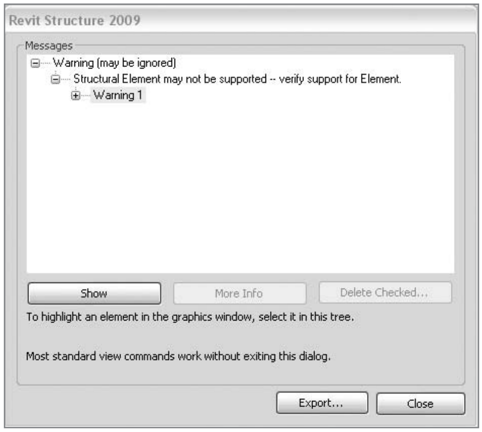


FIGURE 15.14
Check the warning dialog box for unsupported members.

5. Check both Member Supports and Analytical/Physical Model Consistency, and then click OK to close the dialog box.
6. Click Yes to perform the Analytical Model check.

It may appear that Revit did nothing. Choose Tools ➤ Review Warnings. You will see possible model instability warnings. This is a direct result of turning on these settings. Although you cannot correct the situation here, you can select each item and click the Show button, which will give you a view of the offending member. Revit Structure will likely first tell you that there are no good views, but click Show anyway. The more you push the button, the more views of the object Revit Structure will display.

You can save the file to HTML for posting to local networks or Internet sites. Just click the Export button and save to a file.

Tolerances

To understand how the tolerances affect the model, you must first understand the difference between the analytical model and the physical model. We'll return to the topic of tolerances later in the chapter.

Analytical Model

As structural elements are placed into the model, analytical representations of the elements are added as well as a component of each structural element. You may notice that next to each level in the Project Browser there is an analytical level as well. This view does not actually contain additional information; it simply displays different information. Illustrated in Figures 15.15 and 15.16 is the visual difference between an analytical model and a physical model.

There are two sides to every model: the side you are giving to the contractor to bid and build and the side you need in order to analyze the structure. You can create both sides at the same time. Each specific structural member has its specific analytical properties. The next sections will briefly describe these properties, beginning with structural framing.

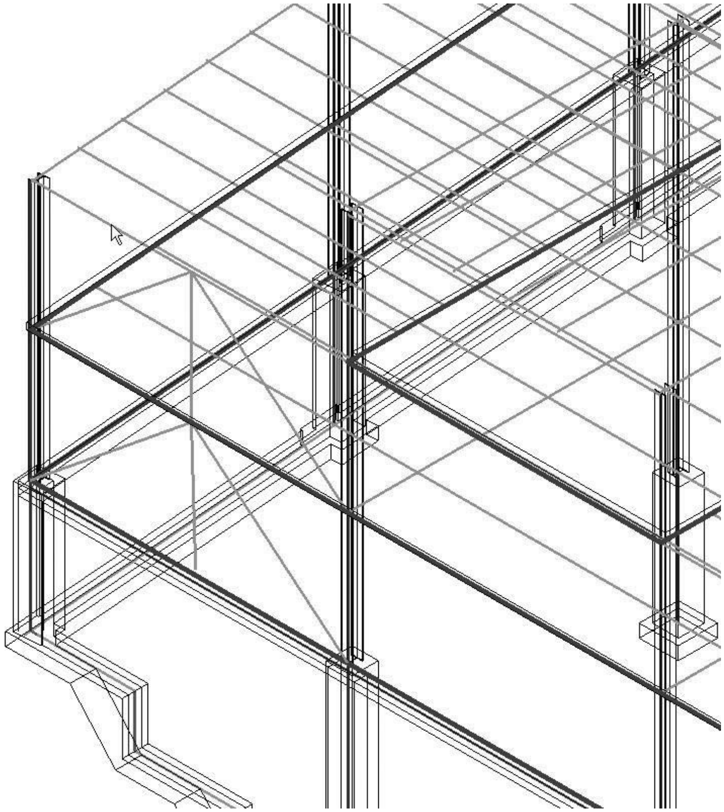


Figure 15.15
The analytic model display

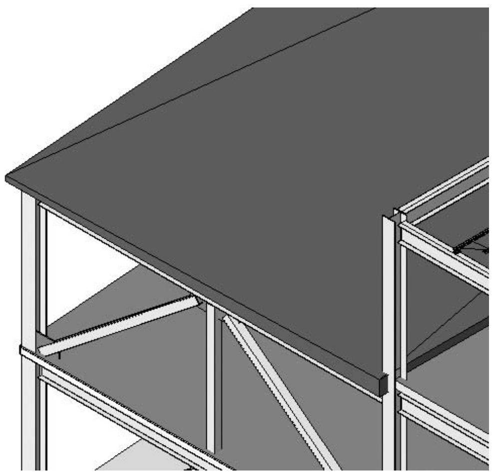
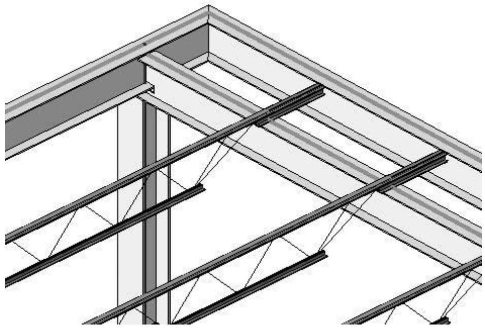


Figure 15.16
The physical model display

Structural Framing

By default, structural framing analytical lines are displayed with an orange line (see Figure 15.17). The analytical plane always stays on the top face of the framing until the framing is attached to another framing member, usually a column. The analytical plane extends into the framing to which a beam is connected.

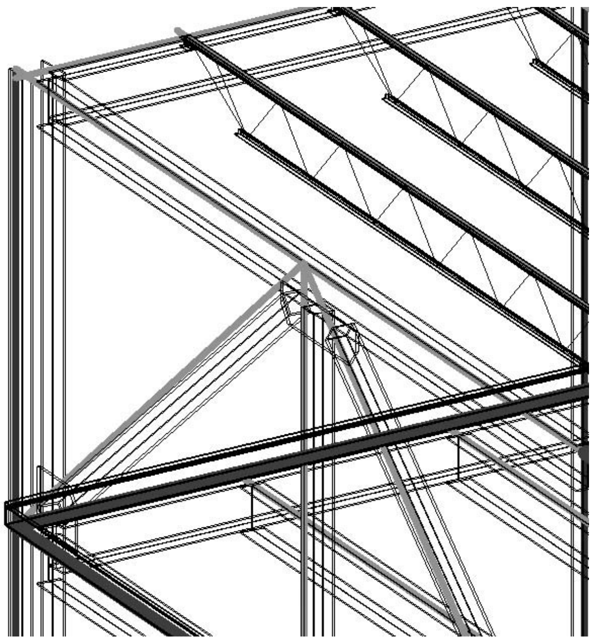
Figure 15.17
The physical and analytical beam representations



For instance, if a beam drops in elevation while attached to a column, the analytical plane will not drop with the actual beam (depending on the method used to drop the beam). It will stay at the level of the adjacent framing. A more extreme case is diagonal bracing (see Figure 15.18), where the framing is at an angle. The framing is being analyzed at a chord center line that extends into the members it is laterally supporting, extending to analytical node points.

A big concern is defining correctly when the physical model stops and the analytical line extends beyond it. This starts to influence the analytical model and can force unwanted inconsistencies.

Figure 15.18
Physical and analytical brace representations at different locations



Next you will see how columns are represented in the analytical model.

COLUMNS

The analytical representation of a column in your project is normally blue and is centered on the column (see Figure 15.19). Columns and beams have some properties in common that have an influence on the plane that hosts them. The procedure is as follows:

FIGURE 15.19
Physical and analytical column representations

1. In `Datasest_1501.rvt` go to 3D View 1 - Analytical if it is not displayed already.
2. Select a concrete column in the basement area.
3. Click the Element Properties button on the Options bar.
4. Scroll down to the Analytical Model settings.
5. Change the Top Vertical Projection setting from Auto-detect to Penthouse (see Figure 15.20) Then click OK.

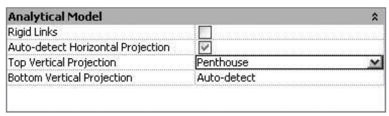


FIGURE 15.20
Changing the Top Vertical Projection setting

You then get a warning (see Figure 15.21) because the analytical projection will be way beyond any acceptable difference between the model and the analytical plane. Granted, some clearance is to be expected, but within reason. This is where the tolerances for the analytical settings come into play.

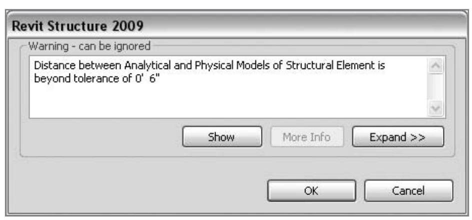


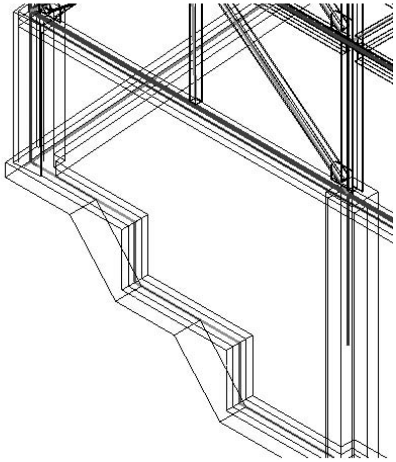
FIGURE 15.21
Distance between analytical and physical models must stay within a defined tolerance.

Next you will examine how wall and footing analytical properties are managed.

Walls and Footings

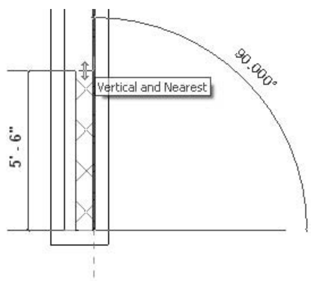
Walls have analytical properties similar to framing (see Figure 15.22). The analytical model can be dependent upon its host geometry, or it can be configured to extend to other members regardless of its host's offset from level. The analytical plane for walls and footings is typically represented in green. To see the analytical properties of a wall, follow these steps:

Figure 15.22
Wall, column, and framing analytical properties have similarities.



1. Find an open area in the Level 1 plan view of the `Datasest_1501.rvt` model.
2. On the Basics tab of the Design bar, draw a structural wall. Make its type Foundation - 12″ Concrete.
3. Set the Top and Base Constraints to Level 1 and T.O. Footing, respectively.
4. Start the Structural Wall command again. Make its type Generic 8″ Masonry.
5. Set Base Constraint to Level 1 and Top Constraint to Level 2.
6. Set the justification for the wall to Finish Face: Exterior.
7. Draw the new wall on top of the 12″ foundation wall with the inside face flush (see Figure 15.23).

Figure 15.23
Coordinating analytical lines in stacked walls



8. Once the wall is drawn, go to the 3D view: View 1 - Analytical. Notice that the analytical plane above is aligned with the analytical plane of the bearing wall below (see Figure 15.24).

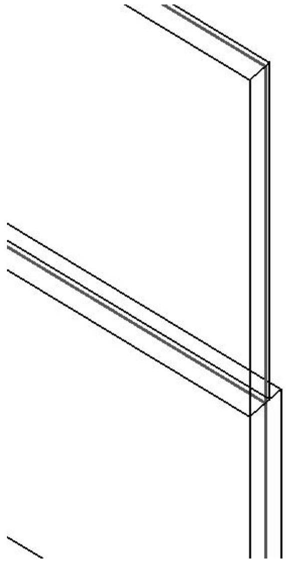


Figure 15.24
Aligning the analytical wall lines for stacked walls of different thickness

9. Select the 8″ wall, and click the Element Properties button on the Options bar.
10. Find the Analytical Model parameters, and browse through the choices. Change the Horizontal Projection value to Exterior Face. Notice that the analytical line changed location but the physical wall did not. You should keep these settings on Auto-detect (see Figure 15.25) and enabled.
11. Change the value for the Horizontal Projection back to Auto-detect.

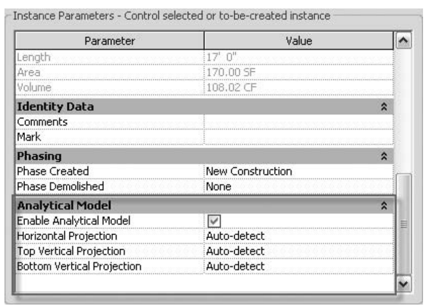


Figure 15.25
Setting the Analytical Model parameters to Auto-detect is the best practice.

The next structural category whose analytical properties we will examine is floors.

Floors

The same analytical functionality that occurrs in beams and walls occurs in floors as well. If a floor is placed in the model, the default analytical positioning will occur at the top outside edge of the extents. The analytical line is typically represented by a brown line (see Figure 15.26).

If the same floor is placed so that it bears on walls below, the analytical plane will adjust to align with the analytical plane settings in the bearing wall. The floor analytical plane will also adjust so that it is positioned at the bottom.

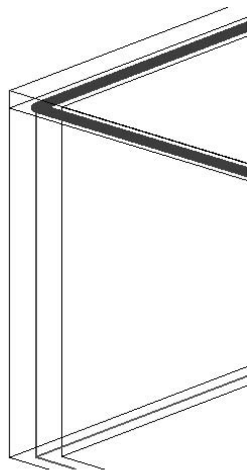


Figure 15.26
Physical and analytical floor representations

The following is a small example illustrating this analytical plane positioning for a wall and floor:

1. Find an open area in the `Datasest_1501.rvt` model using the Level 2 view.
2. Place four Structural 12" concrete walls in a closed rectangular shape from Level 2 to Level 1.
3. On the Modeling menu click Slab.
4. Using the Pick Walls option, pick the four walls.
5. On the Options bar uncheck the option to extend into wall cores.
6. Click Finish Sketch.
7. Answer No to the wall join question.
8. If error checking is still turned on, you will get a warning (see Figure 15.27).

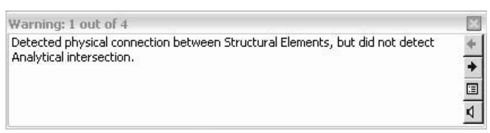


Figure 15.27
Warning of analytical model errors

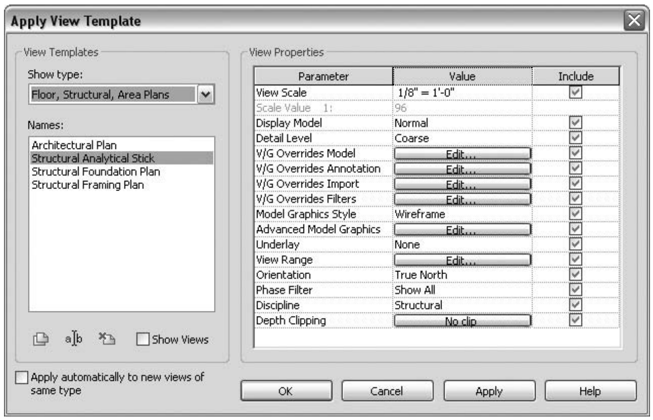
Now go back to the analytical model view and examine the slab and wall analytical lines. This demonstrates the floor and wall interaction. The floor horizontal analytical plane is automatically set at the bottom of the floor and intersects the wall vertical analytical lines.

Creating an Analytical View

Any view can be turned into an analytical view. Here is the procedure:

1. In `Datasest_1501.rvt` right-click the Penthouse structural plan view in the Project Browser.
2. Select Duplicate ➤ Duplicate With Detailing.
3. Rename the new view to **Penthouse - Analytical**.
4. Right-click the new Penthouse - Analytical view.
5. Select Apply View Template.
6. In the Names field, select Structural Analytical Stick (see Figure 15.28).

Figure 15.28
To display the analytical model, use the existing view template.



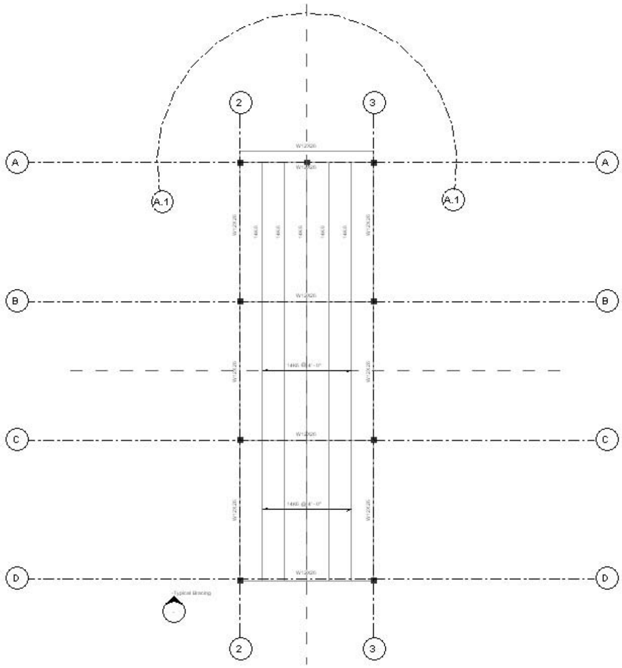
7. You can select only the elements you want applied to the view. In this specific case, *all* of the elements need to be added to the view, so in the View Properties field, all the boxes in the Include column are checked.

Know the Analytical and Physical Models

Now you know that there are two interrelated parts of the model you are keeping track of as you build your virtual model. The settings that control the behavior and the relationship between the physical model and the analytical model should now be more understandable and relevant. It is important to note that as you model a structure, it is good practice to study how your model is coming together analytically as well as physically. The time and discipline it takes to set up a model for analysis will be rewarded when you actually send the model to your analysis software package.

8. Simply click OK at the bottom of the dialog box, and your view should look like that illustrated in Figure 15.29.

FIGURE 15.29
The analytical view display



In the next section on analytical model properties we are finally going to get back to the tolerances functionality that we referred to in the beginning of the chapter.

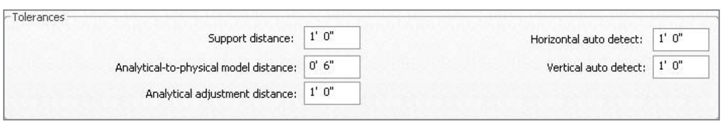
TOLERANCES

Now that you better understand the differences between the analytical model and the physical model, you can begin to learn how tolerances are configured. The following steps will give a short summary of each parameter that you will need to consider when you configure your analytical model:

1. On the menu bar in `Datasest_1501.rvt` click Settings ➤ Structural Settings.
2. Choose the Analytical Model Settings tab.

Under the Tolerances category, there are a few choices listed (see Figure 15.30):

FIGURE 15.30
The Tolerances dialog box



Support Distance This determines how far a cantilever will extend before Revit Structure generates warnings. This is normally set differently depending on the nature of the specific project.

Analytical-to-Physical Model Distance This determines how far past the actual member the analytical line can extend.

Analytical Adjustment Distance This determines how far an Auto Detect analytical plane is allowed to move.

Horizontal and Vertical Auto Detect When a floor bears on walls, it is known that the analytical plane will drop to the bottom of the floor to be in alignment with the top of the wall. If this plane drops (or rises) at an increment greater than 1′-0″, you will need to change the settings to something other than Auto Detect in the object's element properties.

Checks

The remainder of the Analytical Model Settings tab deals with checks that can be enabled so that you will receive timely warnings of possible defects to the analytical model.

Member Support Checks A circular reference will occur when a system of beams frames back to the origin (see Figure 15.31), transferring the load back to the same system rather than transferring the load to a separate bearing member. With this check on, Revit Structure will warn you that you have a circular reference.

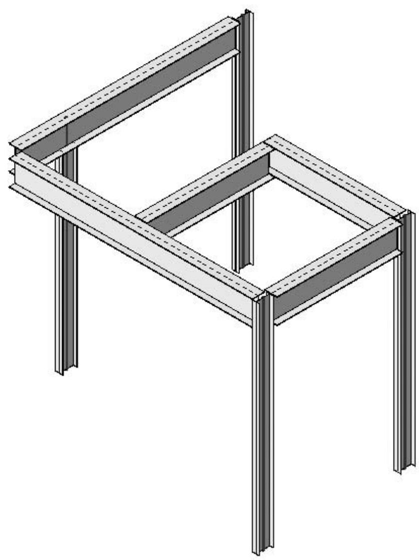


Figure 15.31
A circular reference caused by insufficient bearing of the beam elements

Analytical/Physical Model Consistency Check These settings (see Figure 15.32) are used in conjunction with the automatic checks at the top of the dialog box. When you want to run checks against constructability and structural integrity, these settings will generate a line in the Review Warnings dialog box as you prepare to send your model for analysis.

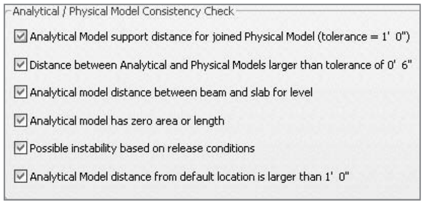


FIGURE 15.32
Set the Consistency Check parameters that will control the warnings reported.

BOUNDARY CONDITIONS

These settings are applicable when defining a condition where other forces are assumed to be in some support of a structural element. A good example of a typical boundary condition is the support of earth underneath a footing or a slab-on-grade. As we place a boundary condition into the model, the appearance in the model will be derived from these settings.

Revit Structure will add a family symbol since the boundary condition is being placed in the model. You should maintain the same family symbol with the actual condition to avoid confusion.

To view the Boundary Condition settings (see Figure 15.33), do the following:

1. Choose Settings ➢ Structural Settings.
2. Choose the Boundary Condition Settings tab. There are four definable boundary conditions: Fixed, Pinned, Roller, and User Defined.

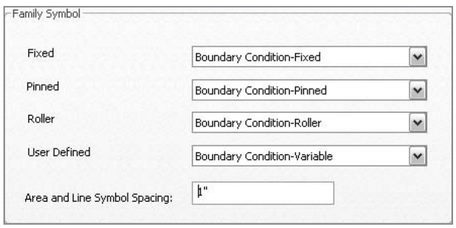


FIGURE 15.33
Setting the analytical boundary conditions

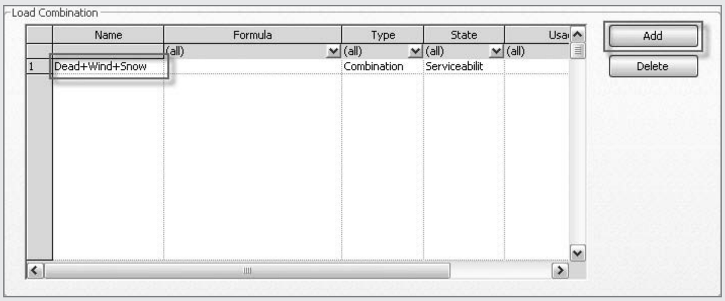
3. Exit the dialog box.

The following exercise will show how to add a boundary condition to your project. The items you are going to select are the orange analytical lines, which indicate the footings. You are specifying a boundary condition based on the footing bearing on earth. Although we do not have topography, this exercise will send the data to your analysis application indicating that there is a natural bearing surface not actually defined in the model.

Exercise: Structural Settings

This exercise is intended to provide you with a comfort level in the configuring of structural settings within your project. The items set here will work in conjunction with the exercise that follows it, which will involve actually placing the loads into the model. **Task 1: Configuring Structural Settings**

1. Open `Datasest_1502.rvt`.
2. Find the T.O. Footing Structural Floor plan in the Project Browser.
3. Right-click and select Duplicate View ➢ Duplicate With Detailing.
4. Right-click the newly created view and select Rename. Call the view **T.O. Footing - Analytical**. Click OK.
5. Right-click the new view and select Apply Template.
6. In the Names category, select Structural Analytical Stick, and click OK.
7. Choose Settings ➢ Structural Settings.
8. Choose the Load Combinations tab.
9. In the Load Combination category, click the Add button.
10. Rename the load condition **Dead+Wind+Snow**, as shown here.

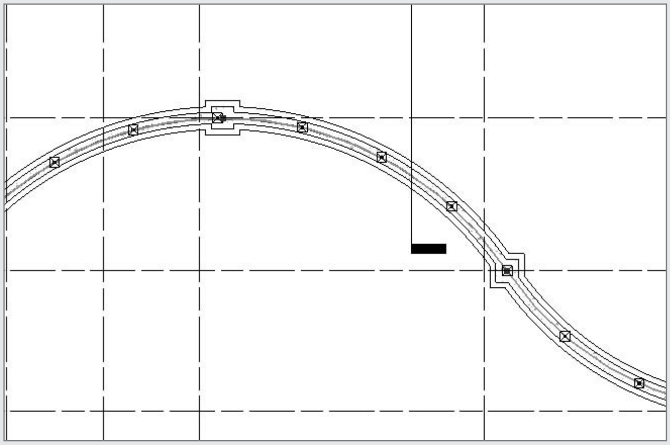


11. Click Add to Load Combination Usage.
12. Name it **Combined Dead and Snow+Wind**.
13. Click the Set check box.
14. Click OK on the Structural Settings dialog box.
15. Save the model if desired.

Task 2: Placing Boundary Conditions

1. Open the T.O. Footing - Analytical plan view if needed.
2. On the Modelling tab of the Design bar, click the Boundary Conditions button.

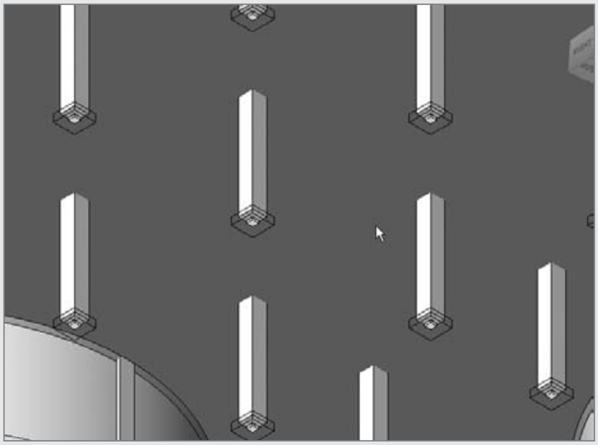
3. On the Options bar, click the Line Boundary Conditions button.
4. For the state choose Fixed.
5. Select the orange analytical lines that define the center of the footings.
6. Proceed by selecting the analytical lines around the entire perimeter, as shown in the following illustration.



7. Choose 3D Views, and select View 1 - Analytical.
8. Start the Boundary Conditions command, and click the Point button on the Options bar.
9. For the state choose Fixed, as illustrated here.

10. Pick the pier footings from underneath. This can get difficult. If it proves to be too difficult, try turning on Model Graphics Style: Shading with Edges and changing the footings to transparent. Do this by:
    - **A.** Right-clicking on one footing.
    - **B.** Choosing Select All Instances.
    - **C.** Right-clicking again and selecting Override Graphics in View ➢ By Element.

D. Choosing the Transparent check box to make the items transparent, as shown here.

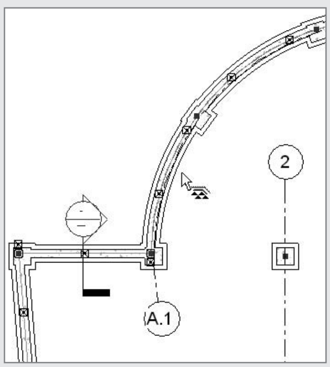


11. Save the model if desired.

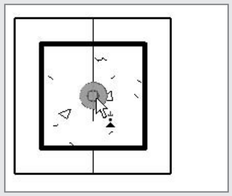
Exercise: Adding a Boundary Condition

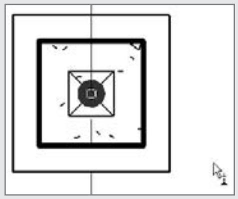
In this exercise you will add an analytical boundary condition to your project.

1. Open the `Datasest_1501.rvt` project file if it is not open already.
2. Right-click the T.O. Footing plan view in the Project Browser, and select Duplicate with Detailing.
3. Rename the new plan to **T.O. Footing - Analytical**.
4. Right-click the T.O. Footing - Analytical plan view, and select Apply View Template.
5. Select Structural Analytical Stick, and click OK.
6. On the Modelling tab of the Design bar, select Boundary Conditions.
7. On the Options bar, click the Line Boundary Conditions button.
8. Select the orange analytical lines in the center of the footings all the way around the building, as shown in the following illustration.

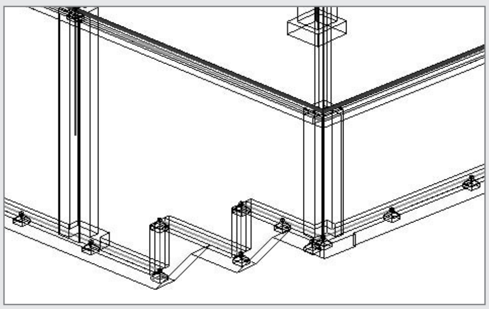


9. Once the entire perimeter is selected, click the Point button from the Options bar, and make sure the State setting is Fixed.
10. Select the blue dots on the spread footings that the piers bear on in the middle of the building, as shown in the following illustrations.





11. Switch to the View 1 - Analytical 3D view to verify, as shown in this graphic.



12. Save the model if you want.

Now it is time to pull all this information together in the following exercise on structural settings.

Placing Loads

There are three different kinds of load placements: Line, Point, and Area. The loads can be placed independently or you can choose a host. That is, the three load placement types can be applied by clicking on the screen to place them or by selecting an object to host them.

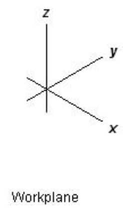
An area load is a good example of choosing a host. If you have a snow load case, it would be a good idea to place it into the model as an area load hosted by the roof or floor object.

To add a load to a model, follow this procedure:

1. Open the file `Dataset_1501.rvt`.
2. Make sure you are in 3D Views, and choose View 1 - Analytical.
3. On the Modelling tab of the Design bar, click the Loads button.

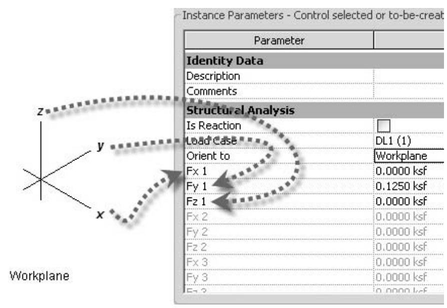
The first thing to observe is the new icon that is placed within the view window (see Figure 15.34). This is your guide as you set the properties of the direction to which the load is reacting. This icon indicates the work plane, and it is directly related to the load you are about to place.

FIGURE 15.34
Directional guide as you place your load



1. Select the Area Load With Host icon from the Options bar.
2. Click the Element Properties button on the Options bar. You will see the following properties:
   - Load Case can be applied to the load you are physically adding to the model.
   - Orient To specifies if you are in an orientation to the work plane (see the workplane icon at the bottom left of your drawing area).
   - Fx, Fy, Fz indicate the direction the load is coming from (see Figure 15.35). Again, you can tell by looking at the new icon in the view window which item needs to contain a value.

FIGURE 15.35
Directional load indications



We want to place an area load with a load case of DL1(1) on the right side of the foundation.

Looking at the x, y, z icon, we know that Fz1 seems like it should be blank; however, once we place the load, the work plane will orient to reflect the angled wall.

1. For Fz1, type -.**1000** (notice, this is a negative value) and then click OK.
2. In the View 1 - Analytical view, select the basement foundation wall to the right of the model. Once it is created, select the new load, and notice that the Host work plane is now oriented to reflect a z value at the skewed angle of the wall, as illustrated in Figures 15.36 and 15.37.

FIGURE 15.36
Note the orientation of the work plane

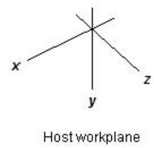
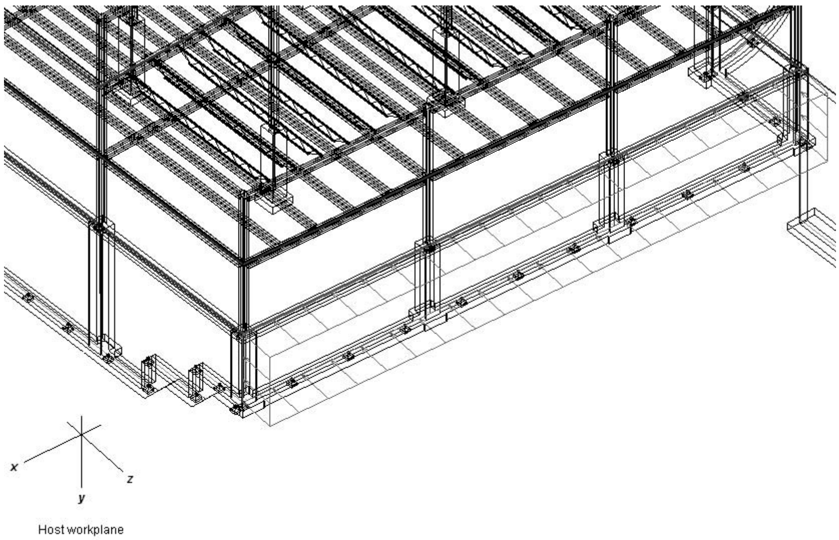


FIGURE 15.37
Note the orientation of the work plane



Leave the model open for the next section, where you will add a line load to your project.

Line Load with Host

Adding a line load with host is a little less confusing than adding an area load. Since a line load has no z direction whatsoever, the initial host work plane will point the force based on 0, 90, 180, and 270 degrees. For an example of how to place a line load with a host, we will be using the roof plan. The load will be placed laterally along the west side of the roof line.

1. In the Project Browser, right-click the Roof plan view, and select Duplicate View ➢ Duplicate with Detailing.

2. Rename the new copy **Roof - Analytical**.
3. Right-click Roof - Analytical, and select Apply View Template.
4. Choose Structural Analytical Stick from the Names category; then click OK.
5. If you have previously placed a boundary condition on the footing, it may be visible. If it is, leave it on.
6. On the Modelling tab of the Design bar, click the Loads button.
7. On the Options bar, select Line Load with Host.
8. Click the Element Properties button.
9. Change Fx 1 to **1.0000**, and Fy 1 and Fz 1 should be 0.0000 (see Figure 15.38). Click OK to exit the dialog box.

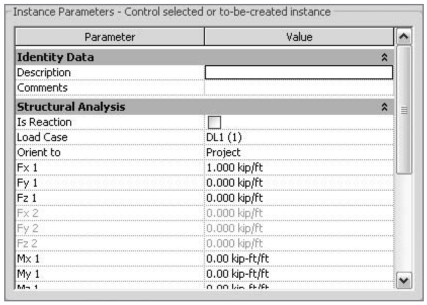


FIGURE 15.38
Setting analytical instance parameters

10. Click the orange analytical planes along column line 1 (see Figure 15.39).

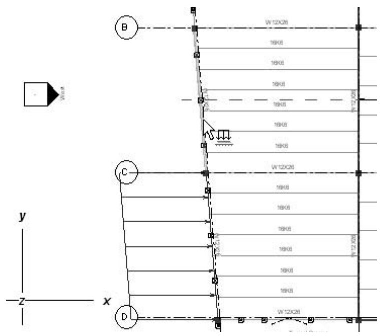


FIGURE 15.39
Selecting the analytical lines

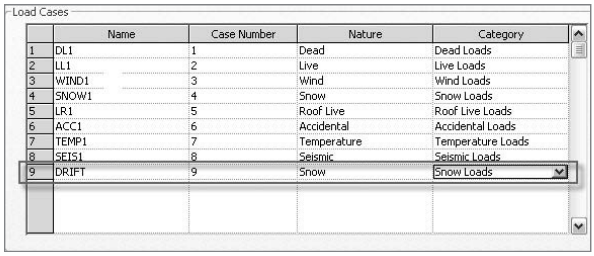
11. Once the west column line is done, right-click and choose Cancel to end the command.

The next section will add an area load to your project.

Area Load (By Sketch)

An area load by sketch allows the designer to draw an area where a concentrated load will occur. A good example would be a drifting snow load. Normally in this situation it is a good idea to create a new load case as well.

1. Go to Settings ➢ Structural Settings ➢ Load Cases.
2. Click the Add button, and rename the new load case **DRIFT** (see Figure 15.40).
3. Set the parameters as follows: the Nature is Snow, and the Category is Snow Loads. Click OK to exit the dialog box.



	Name	Case Number	Nature	Category
1	DL1	1	Dead	Dead Loads
2	LL1	2	Live	Live Loads
3	WIND1	3	Wind	Wind Loads
4	SNOW1	4	Snow	Snow Loads
5	LR1	5	Roof Live	Roof Live Loads
6	ACC1	6	Accidental	Accidental Loads
7	TEMP1	7	Temperature	Temperature Loads
8	SEIS1	8	Seismic	Seismic Loads
9	DRIFT	9	Snow	Snow Loads

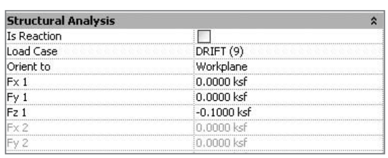
FIGURE 15.40
Adding an area load case

4. Go to the Roof - Analytical plan if needed.
5. On the Modelling tab of the Design bar, click the Loads button.
6. On the Options bar, click Area Load, as shown in Figure 15.41.

FIGURE 15.41
Clicking the Area Load button on the Options bar

Notice that for this command, the Design bar turns into a Sketch bar.

7. Click Load Properties.
8. Change the Load Case to DRIFT (9).
9. Change the Fz 1 value to **-.1000 ksf** (negative). All other F values are 0.0000 (see Figure 15.42). Click OK to exit the dialog box.



Structural Analysis	
Is Reaction	☐
Load Case	DRIFT (9)
Orient to	Workplane
Fx 1	0.0000 ksf
Fy 1	0.0000 ksf
Fz 1	-0.1000 ksf
Fx 2	0.0000 ksf
Fy 2	0.0000 ksf

FIGURE 15.42
Setting the F values for analysis

10. Draw an area about 8′-0″ wide that abuts the framing leading up to the penthouse.
11. Mirror the area to the opposite side of the penthouse (see Figures 15.43 and 15.44), and then click Finish Sketch.
12. Save the model if desired.

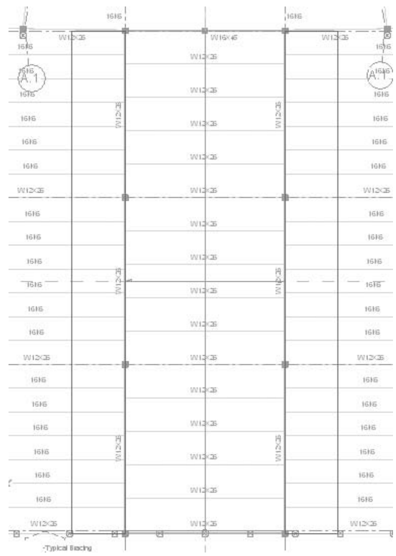


FIGURE 15.43
Placing the area load in sketch mode

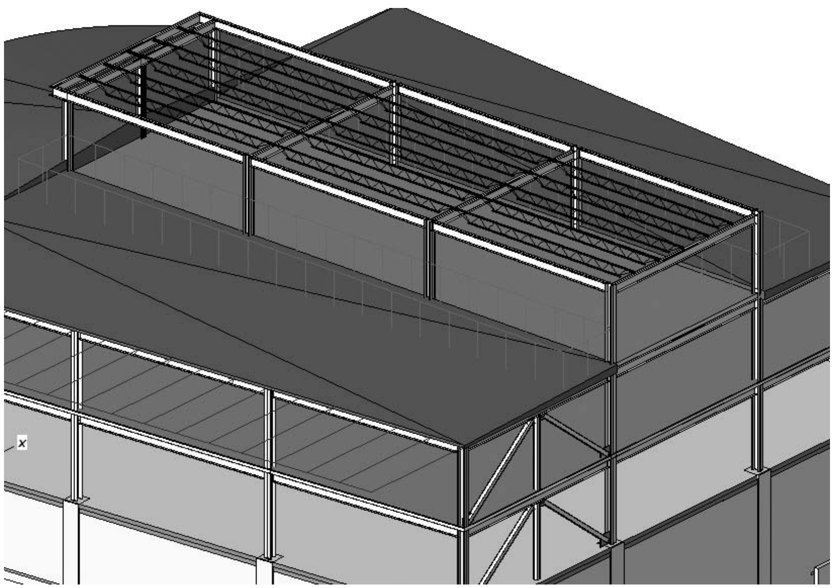


FIGURE 15.44
Loads adjacent to the penthouse in 3D view

Exercise: Placing Loads

In this exercise you will have three tasks to accomplish: you will create load cases, draw up an analytical plan, and place loads onto the roof of the project.

Task 1: Create a Load Case

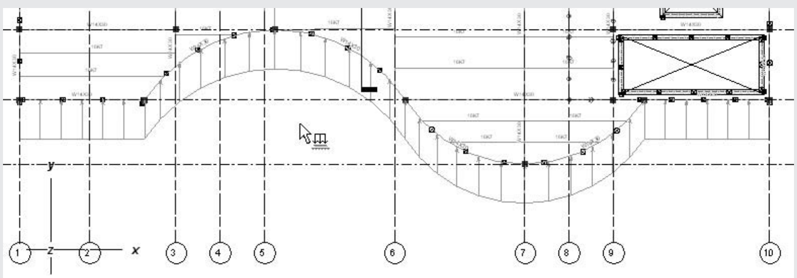
1. Open the file called `Dataset_1502.rvt`.
2. Choose Settings ➢ Structural Settings ➢ Load Cases.
3. Click the Add button, and call the new load case **DRIFT**.
4. Set Nature to Snow and Category to Snow Loads; then click OK to exit.

Task 2: Create an Analytical Roof Plan

1. In the Project Browser, right-click 14 ROOF.
2. Choose Duplicate View ➢ Duplicate with Detailing.
3. Rename it **14 ROOF - analytical**.
4. Right-click 14 ROOF - analytical, and select Apply View Template.
5. Choose Structural Analytical Stick. Click OK to exit.

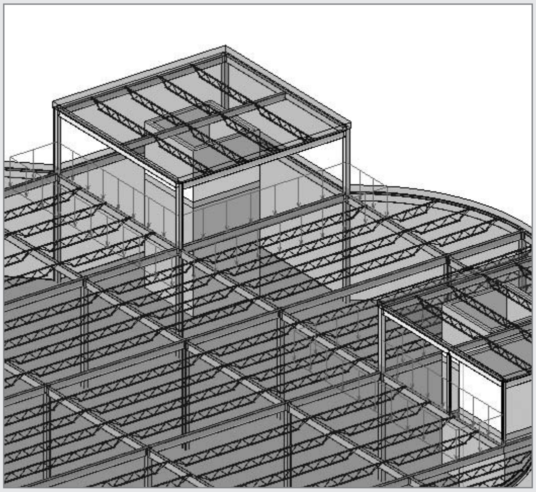
Task 3: Place Loads

1. Open the 14 Roof - analytical plan if needed.
2. Select Loads from the Modelling tab of the Design bar.
3. On the Options bar, select Area Load.
4. Draw a drift region around the two higher penthouse areas, about 8´-0˝ away from the penthouse. You can also click the Pick Lines button to aid in the drawing of the loads.
5. Click the Load Properties button.
6. Under Structural Analysis, change the load case to DRIFT (9).
7. Change Fz 1 to **-0.1000**.
8. Click OK, and then click Finish Sketch. Go back to View 1 – Analytical; it should now look like the following graphic.



9. In the Modelling tab of the Design bar, click the Loads button again.
10. This time you are going to place a line load by host on the south edge of the building. Click the Line Load With Host button.

11. Click the Element Properties button.
12. Set the Load case to WIND1 (3) if needed.
13. Set the Fy 1 to **0.500 kip/ft**; then click OK.
14. Using the 14 Roof - analytical plan view, click the analytical beam lines on the south edge of the building, as in the following graphic.



15. Save the model if desired.

Exporting for Structural Analysis

With the loads in place, the Revit Structure model is now ready to be exported. You have done everything you can to add information that will allow your analysis application to take the model the rest of the way.

Out-of-the-box Revit Structure 2009 does not come with a link to any analysis software packages. It is up to the analysis vendor to provide links and enablers for its applications to establish the relationship between Revit Structure and its products. All of the major vendors do have this link available, fortunately. The first thing you need to do is either contact your analysis software provider, or simply go to its website and do a search for "Revit Structure." Some sites are easier than others to use to find the various components, but once you do find them, they are usually free of charge.

After downloading the integration link software from your analysis vendor, you will need to install it, as shown in Figure 15.45. The procedure to export your model to the software is as follows:

1. Make sure the applications have been loaded, and then open Revit Structure.
2. Go to Tools ➤ External Tools ➤ Send Model To ➤ *Name of analysis program* (see Figure 15.46).

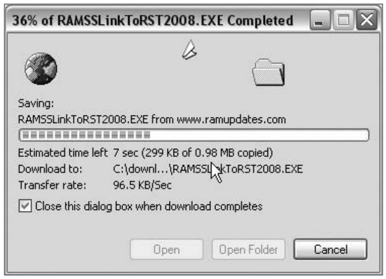


FIGURE 15.45
The Bentley Ram link

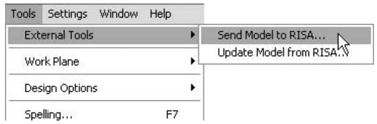


FIGURE 15.46
Sending the model to RISA

3. The application's dialog box then will be displayed (see Figure 15.47). You will be given some choices as to:
   - ◆ The extent of elements that you are exporting
   - ◆ Specific application questions you will need to answer for that particular analysis package
4. Depending on the application, Revit Structure will either export a file in that program's native format or create a `.bim` file. Some applications will read this, while others will not.

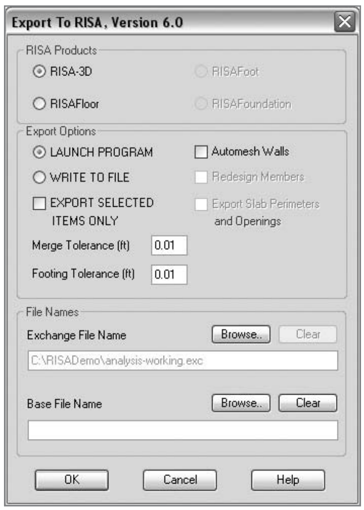


FIGURE 15.47
Export options in the RISA analytical package

Real World Scenario

The Viability of Analytical Model Integration

This part of Revit Structure is constantly evolving as the AEC industry and BIM technologies mature and use ever-more-powerful computing tools. It's an incredibly complex notion to combine and coordinate analytical and physical models all in one file. At this moment the promise of integrating Revit Structure with analysis links has not proven itself totally in practice. Numerous difficulties may occur that will deter you from using it as intended.

But the situation seems to be changing even as this book is being published. With its recent purchase of Robobat, Autodesk is poised to incorporate its own analysis software into its software suite, or so it would appear. Already with the Revit Structure Extensions, analytical capabilities are being nicely integrated into Revit Structure. We are bound to see dramatic changes in the near future as the software makers contend with the ever-expanding reality of BIM design.

Autodesk also has just recently entered into an agreement with Bentley to coordinate their efforts more and to better integrate products such as RAM Concept with Revit Structure. The importance of this agreement cannot be understated, since for several years there has been fierce competition between these two AEC software giants for control of the BIM market. It got so bad that you could not even find the integration link or any mention of Revit on the Bentley web site. It was hidden away and took much scrounging around to find it. Whether or not this agreement will succeed could well foretell the success of a more open source approach to BIM software development. This is not a truly open source agreement, but since these two companies control an overwhelming market share, in practical terms it could have a great influence on that happening.

At this time it must be said that analysis integration is very challenging to undertake in your project, which is more the reason why collaboration between Revit Structure and its analysis partners needs to be strengthened. The idea that you can import and export the model through the analysis package multiple times through the entire design period is for the most part still theory. Very few structural design firms have been able to accomplish that goal. The reasons are many and varied and quite dependent on which analysis package you are using. Each has its own problems.

Your best bet to get started in using this functionality is to use it for preliminary analysis of your structure. Work on one import or export of data as a starting point and try to make that work in a project. Start your model in the analysis package and import it into Revit Structure or vice versa. That in itself will save valuable time and should not be ignored.

But keep a close eye on upcoming events since this story is changing almost daily. At some point the theoretical vision of integrating analytical and physical model activity will be achieved and will create a very dynamic flow of structural design data. Hopefully that goal will be achieved sooner rather than later, since the efficiencies to be gained are quite substantial.

Transfer of Data

The good thing about this method of importing and exporting is that regardless of the analysis application to which Revit Structure is linked, it is merely an exchange of data, not a physical model. This is important because there is no worry about insertion points or items that did not get selected.

To import a model back into Revit, or to import a model that may have started in an analysis application, you follow the same procedure as you did to export the model.

1. Choose Tools ➤ External Tools ➤ Update Model From *Application*.
2. The application will then prompt you to make some choices based on geometry that is not loaded into the Revit Structure model at the time of import (see Figure 15.48).

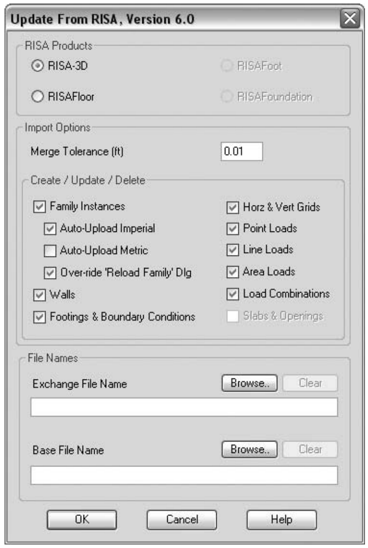


FIGURE 15.48
Updating the model from the RISA analysis application

3. At this point, you can click the Browse button and find the model that you wish to import into Revit Structure.

This has been a brief glimpse at this difficult subject, showing how the analytical model can be exported and imported into Revit Structure.

The Bottom Line

Configure Revit Structure structural settings and create loads for your project. The Structural Settings dialog box contains the tabs that will allow you to configure loads for your project. Load Cases, Load Natures, Load Combinations, and their usage form the basis for preparing your analytical model for export to analysis software.

Master it

1. True or False: Revit Structure has the ability to perform structural analysis.
2. True or False: Revit Structure cannot combine load cases.
3. True or False: A circular reference will occur when a system of beams frames back to the origin.

4. True or False: A good example of a typical boundary condition is the support of earth underneath a footing or a slab on grade.
5. Where are the settings located that allow you to turn on consistency checking?

Place analytical load patterns onto your model. Loads are placed in the model in anticipation of using them for preliminary analysis. Several placement methods are possible within Revit Structure. Each of these methods can be applied in two ways.

Master it

1. Name the three different kinds of load placements.
2. The analytical properties of an element can depend on one of two things. What are they?
3. True or False: The directional guide is an icon that indicates the work plane for the load you are about to place.
4. What two ways can load placements be applied?
5. True or False: Adding a line load with host is a little less confusing than adding an area load since a line load has no z direction whatsoever.

Import and export your virtual model from Revit Structure to structural analysis software. Once the loading is created and placed, the model is ready to be exported to an analysis application. Once the analysis is complete, it can then be imported back in to Revit Structure and will automatically update the model.

Master It

1. True or False: The integration links come prepackaged with Revit Structure.
2. When the application's dialog box is displayed, you will be given some choices. To what do they refer?
3. What type of file will Revit Structure export to the analysis application?
4. True or False: Importing and exporting to the analysis application exchanges the physical model back and forth between Revit Structure and the analysis application.

Part 5

Advanced Topics

Chapter 16

Project Phases and Design Options

To appreciate the power of Revit Structure in working through the design process of a typical building project, you must improve your skills in the basics to the point where you start to feel comfortable using project phases and design options. As you will see in this chapter, you can manipulate the project data to describe multiple building scenarios for your design—and all in one Revit Structure file. Many people tend to shy away from using project phases and design options at first. But once you learn how they work and the logic behind their use, you will find yourself employing them more and more. Using these features is an efficient way to develop your model.

In this chapter you will examine the basic commands and rules for phases and design options, and learn how they can be applied to your projects. We'll introduce several real-world situations to illustrate their application.

In this chapter you will learn to:

- Create project phases to manage element assignments
- Display project phases in your project views
- Create design options to manage element assignments
- Display design options in your project views

Working with Project Phases

Working with project phases is a task that many projects will require. Revit Structure has a good system for you to create and manage them. What kind of phases? A phase could involve the objects in your model that belong to an existing building to which you are adding a new addition. Or it could involve the items that are being demolished. Each can be identified as a phase. You can create your own phases or use the default ones.

You assign elements in your model to an appropriate phase. You then create views that display the phases accordingly. It sounds simple enough, but you must pay close attention to your element and view properties when you are working with phases.

Probably the most common application of phasing will be in cases where an existing building is related to your new design. You have two basic options for setting up this type of project environment: Either create the existing model and new model together in one file, or create the existing model in a separate file and link it to your new model. Depending on the particulars of the project you could decide to go either way and both are appropriate.

Construction sequencing of the design model also uses phasing to great advantage. Once you have built the model, you can reassign the elements to different phases that match the stages of the actual construction. You can isolate those elements for display in construction views and in schedules. The BIM model starts to become a rich resource of information for all types of purposes. Demolition drawings are another obvious use of phasing.

Figure 16.1 shows a view of the Williams-Drummond House historical remodel project where only the existing structure is displayed. In this project, historical preservation was an important factor for the owners. The design ideas for the residence received close scrutiny from the local building department and historical committee of the city. Figure 16.2 shows the same house design in its final completed stage after the remodel.

As you will see later in this chapter, various design options were created to show several completed remodeling ideas. These were then documented and presented in the early part of the project to the building department and historical committee for review. The phasing for new and existing elements was done within the various design options presented. We will return to this example later.

FIGURE 16.1
The existing historic residence

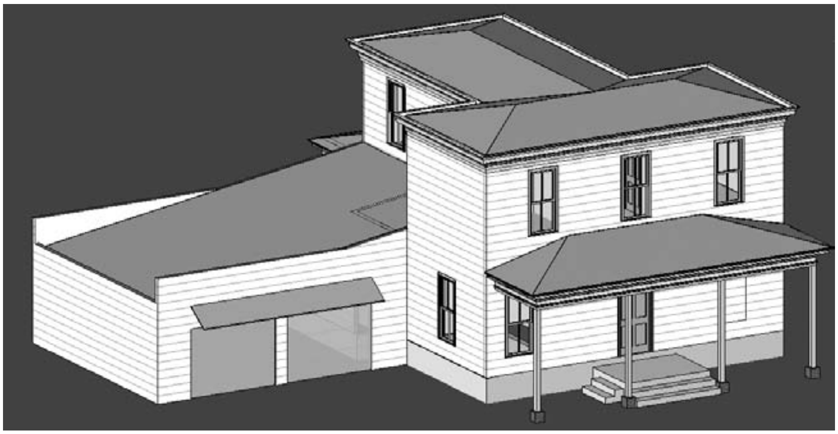
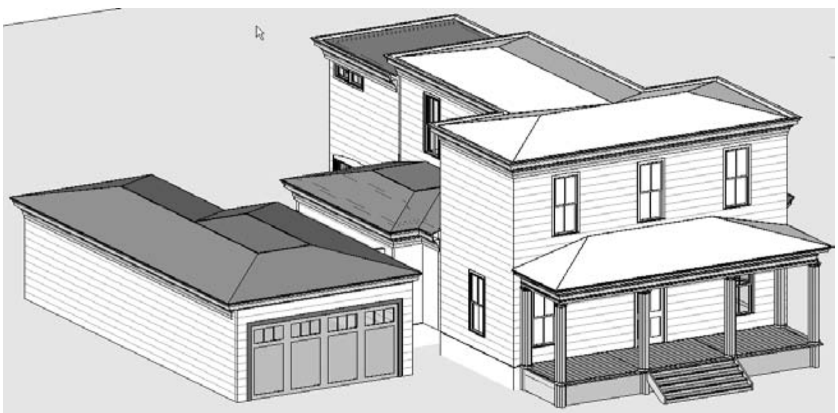


FIGURE 16.2
A new construction option for the residence



The Properties of Phases

Project phasing basically means that structural elements within the scope of your design are categorized and ordered in such a way that identifies their chronological building order or some other organizational task. Revit Structure allows you to organize and manage the many types of design and construction tasks that you may encounter. As the BIM approach matures and becomes normal practice, you will no doubt be faced with this task more and more.

Creating Phases in Your Projects

When working on a project, you will need to consider how to *phase*, or make into discrete portions, the segments of the design or construction process you are documenting. Revit Structure helps you by assigning the elements you introduce in your model to specific phases. The remodel of an existing building will have two basic phases—existing construction and new construction—as well as a phase for demolished elements. The new construction can be broken down again into subphases.

For example, remodel work in an existing hospital may require moving staff and machinery out of the way and then back again during the course of a project. All that would have to be phased correctly to allow for the continuation of normal operations in the overall hospital. So in your model you develop views that show the various conditions and you assign to elements in the model an identification relevant to the phases you create.

Project Phases

The Phase dialog box is accessed on the menu bar under Settings ➢ Phases…. There are three basic areas in the dialog box that are used to create and configure your project phases that you need to study and understand to get started. Those are accessed with the tabs: Project Phases, Phase Filters, and Graphic Overrides.

There are two default project phases, New Construction and Existing (see Figure 16.3), although you can add as many as you like. To add a new phase to your project simply click the Before or After button in the Insert area to place it into the correct sequence.

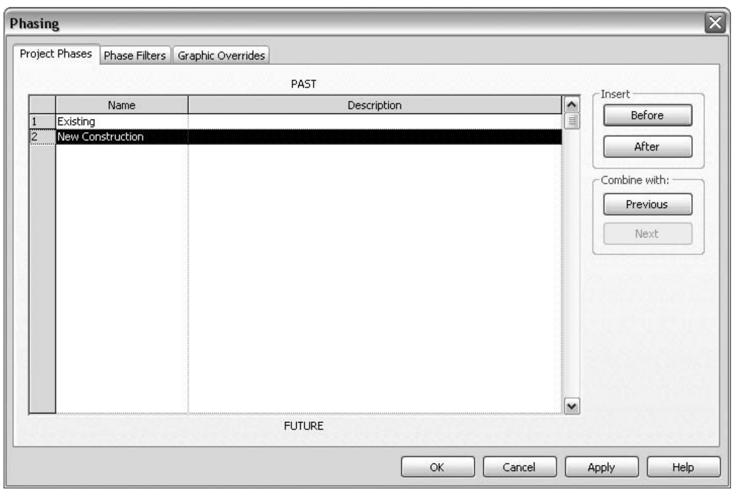


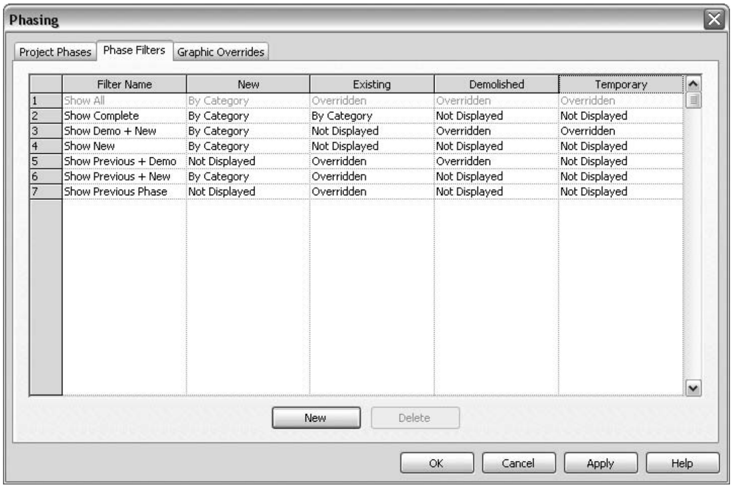
Figure 16.3
The Phasing dialog box

Phase Filters

Each phase that you use has associated filters that control how elements are displayed in any particular view where that phase is enabled (see Figure 16.4). Each view in your project has parameters that let you set a phase and apply a phase filter for that view. Setting these options in the View Display dialog box controls what is shown in that view. Each element also has phasing controls. When these options are set in conjunction with the view controls, it creates what is referred to as the *phase status* for the element.

For instance, if the phase is set to New Construction in your view, what might you want to display? You probably just want the new work shown. By setting Phase Filter to Show New, you create a phase status for elements so that:

- The model elements defined as New Construction will be displayed By Category.
- Existing, Demolished, and Temporary items will not be displayed.



	Filter Name	New	Existing	Demolished	Temporary
1	Show All	By Category	Overridden	Overridden	Overridden
2	Show Complete	By Category	By Category	Not Displayed	Not Displayed
3	Show Demo + New	By Category	Not Displayed	Overridden	Overridden
4	Show New	By Category	Not Displayed	Not Displayed	Not Displayed
5	Show Previous + Demo	Not Displayed	Overridden	Overridden	Not Displayed
6	Show Previous + New	By Category	Overridden	Not Displayed	Not Displayed
7	Show Previous Phase	Not Displayed	Overridden	Not Displayed	Not Displayed

Figure 16.4
Applying filters in the Phasing dialog box

You apply phases and phase filters in each view by setting parameters in the View Properties dialog box. More on this subject of assigning phases to elements will be presented in greater depth in the section "Assigning Elements to Phases."

There are six default phase filters that you can use immediately, and others can be created as needed. The default configuration of any phase filter can also be changed to suit your needs.

Suppose you have a phase set as New Construction, with only the Existing phase preceding it. The following list explains each filter associated with this phase:

Show All This filter displays all elements, including existing, demolished, and temporary elements.

Show Demo + New This filter displays demolished elements and all of the new elements that you add to your model.

Show New This filter displays only the new elements that you have added to your model.

Show Previous + Demo This filter displays all elements from phases before the current, demolished, and temporary elements.

Show Previous + New This filter displays all elements from the previous phase that were not demolished, as well as all added new elements.

Show Previous Phase This filter shows all elements from the previous phase.

Note that if you are in the first phase there is no previous phase, so if you enable the Previous Phase filter, nothing will be displayed.

The phase filter categories include:

Not Displayed The elements will not be displayed in that phase.

By Category An element will be displayed as it is defined in the Visibility Graphics dialog box of each view.

Overridden The graphic display is overridden for those elements in each view.

You will most likely display new elements By Category. At the same time, you may want existing elements to appear shaded or with a particular line style so they are easily discernible in comparison to the new work. Demolished items as well may be best displayed in a common way, such as the default, which is visually shown as red and a hidden line pattern. The next section will show you how to use the graphic overrides and configure them to your needs.

To create a new phase filter, simply click New in the Phase Filter tab then provide an appropriate name for that filter. Any number of phase filters can be created and configured in your project.

Phase Overrides

Phase overrides allow you to modify the display of elements in each phase filter to something other than By Category. Figure 16.5 shows the Graphic Overrides tab, which controls assignments of the various line styles, patterns, and materials within your project views. The overrides work like this:

- Line styles and patterns can be altered for all for elements that are in projection or that are surfaces in projection.
- Line styles and patterns can be altered for all for elements that are cut by the view plane.
- The Halftone check box will tone the elements.
- The Material display of different phase statuses can be configured and applied universally.

All of these phases, filters, and overrides seem a bit daunting at first, but do not be scared off from learning to use them since they will help coordinate your project efforts tremendously. Start with a simple case and get that to work. Later you can move to more complex uses.

Now that you have created the phase and configured the filters to your liking, you can move on to assigning the elements in your model to a phase. This assignment will be an instance parameter setting for each element.

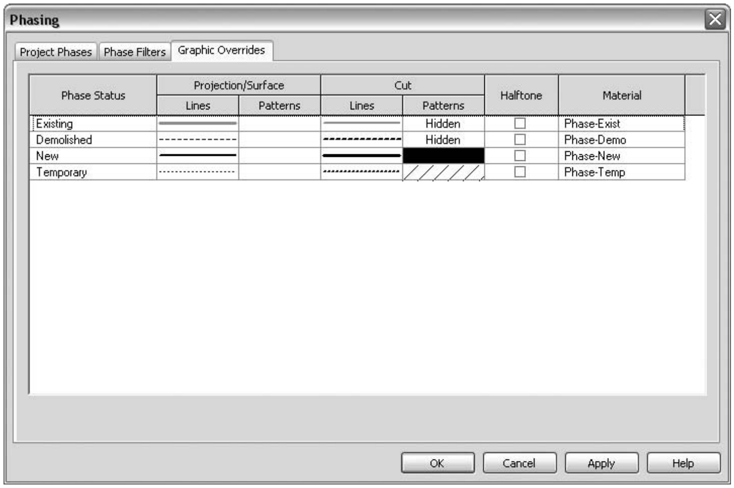


FIGURE 16.5
The Graphic Overrides tab in the Phasing dialog box

Assigning Elements to Phases

In the default setup, you configure the Phase Created and Phase Demolished options for an element to create a phase status for that element. The phase status can be:

Existing This refers to an existing element in the project.

New This refers to a new element in the project.

Demolished This refers to an existing element in one phase that is demolished in a subsequent phase of the project.

Temporary This refers to an element that is created and demolished in the same phase of the project.

In your project you will assign each element to a phase. Each element has two phase parameters: Phase Created and Phase Demolished. The default value for Phase Created is New Construction, and the default for Phase Demolished is None. It makes sense that a new element is not demolished (see Figure 16.6).

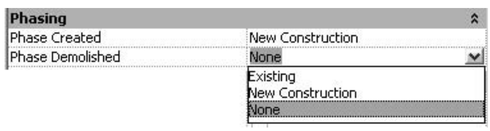
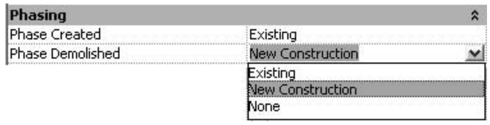


FIGURE 16.6
Phase parameters for a new element

ELEMENT DEMOLITION

For an existing element, set the Phasing Demolished parameter to the phase in which you want it to be demolished (see Figure 16.7).

FIGURE 16.7
Phase parameters for a demolished element

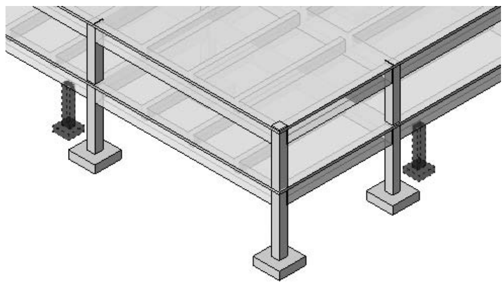


The Demolish tool is found on the Tools toolbar (appropriately marked with a hammer icon) and is used to mark elements as demolished in the current phase. Demolishing an element in one view will result in it being marked as demolished in all views that have the same phase.

TEMPORARY ELEMENTS

Temporary elements are those that are created and demolished all in the same phase, such as temporary shoring (see Figure 16.8). For a temporary element, simply set the Phase Created and Phase Demolished parameters to the same phase.

FIGURE 16.8
Temporary shoring (shaded and hidden line) supporting new concrete



Our discussion so far has centered on elements that are all in one file. But many times you must create and maintain phasing for linked models. Let's take a look at how that works.

Linked Revit Structure Models and Phasing

Another important consideration to how you approach your project phasing is whether you are linking other Revit Structure files into your project. For instance, how would you approach a project with an existing building that is being remodeled? There are two obvious approaches:

- Create one Revit Structure file that incorporates the existing structure and the new remodel work that has to be done.
- Create a Revit Structure file for the existing structure and link that file into a file that contains only the new work.

Depending on the project, a case can be made for going either way. You may find that linking in the existing building will give you greater flexibility. One important reason for this approach

is that computer performance could be improved by being able to unload it when you do not need it displayed in your views. It might also be easier with multiple people working on the project if there are several files on which they can work. One potential challenge, though, is that controlling the phasing through the link can be a little difficult.

With a linked model, you control the phase and phase filters in the RVT Link Display Settings dialog box, as shown in Figure 16.9, which you access through the Visibility Graphics dialog box. You can set the phase settings for a particular view from the host view file or the linked view file, or employ a custom approach that uses settings from each file. Be careful though, because things can get somewhat confusing when you start altering the settings. When you get the settings the way you want them, create a view template that you can apply in similar circumstances.

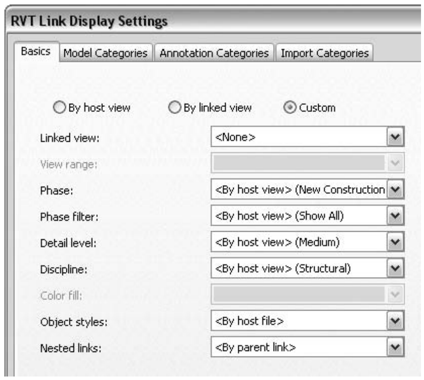


Figure 16.9
Control the phase of the linked file using the RVT Link Display Settings dialog box.

Next you will see how phases can be used to your great advantage to display a complex situation within a project. Then you will do an exercise to demonstrate how all this goes together.

Displaying Phases in Views

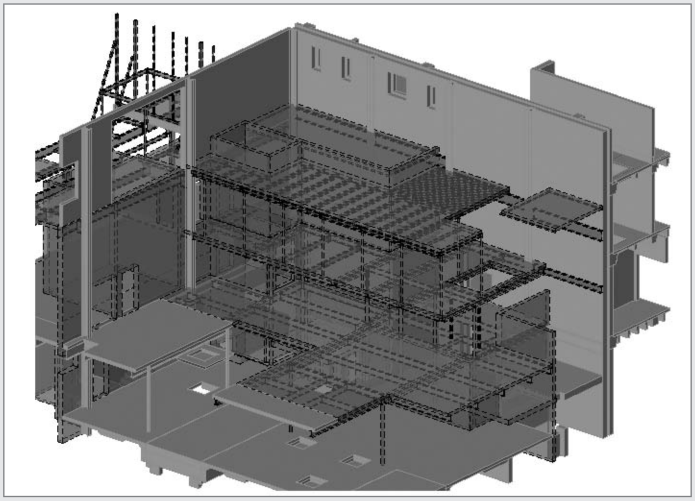
First you learned how basic phases are created. Then you learned how to assign to your model elements the phase in which they occur. Now let's look at how a view is configured to display elements in a particular phase. We'll consider two examples that illustrate both ways of working.

Real World Scenario

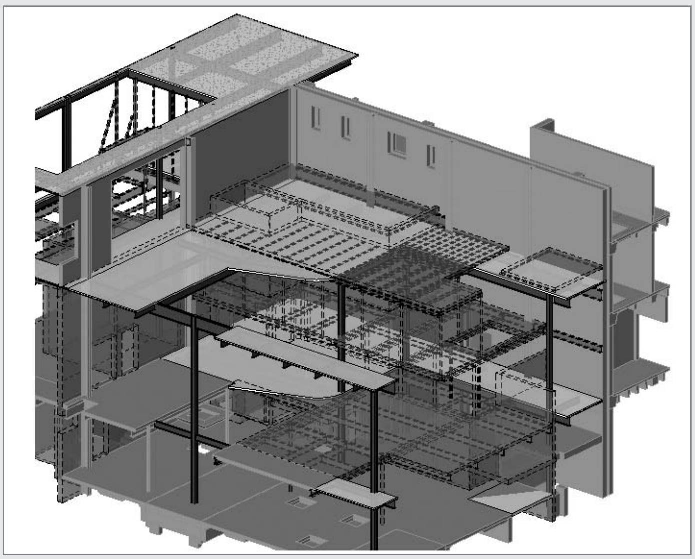
Using Phasing in a Difficult Design Situation

The project examined in this scenario was a remodel of a large existing industrial building. The project model had two files, with the existing structure linked into the new structure. A large amount of demolition had to be documented. With one corner of the existing structure in particular, understanding in two dimensions what needed to be demolished in relation to the new work was quite difficult. By using three-dimensional cutaway views that showed the phasing of that area, we ensured that the challenge of the remodel came more clearly into perspective.

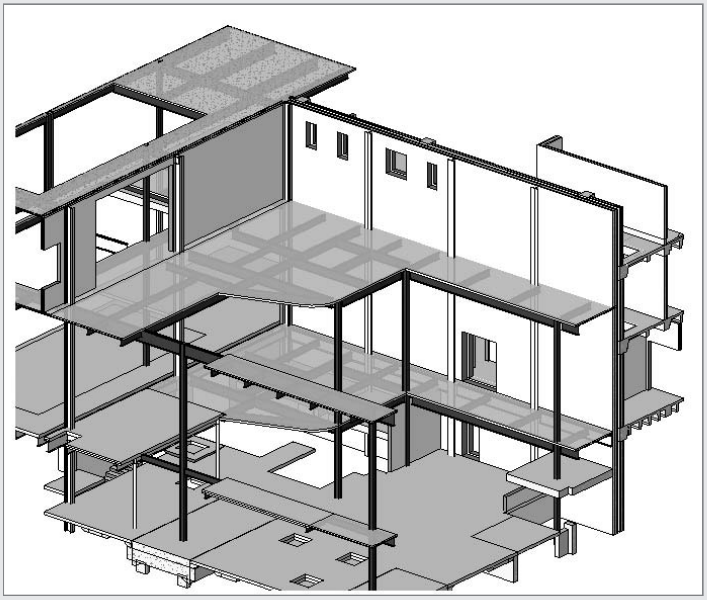
First, we created a view that showed the existing and demolished model elements, as shown in the following illustration. The demolished elements appear in red and as hidden lines. The existing elements that are remaining are shown gray and toned.



We then created a second view from the same orientation and adjusted the phasing for that view to show all elements regardless of status. So in this case the view displayed new, existing, demolished, and temporary elements, as shown in the following graphic. The Phase option was set to New Construction and the Phase Filter set to Show All. This view really helped since there were numerous stacked tube columns and lots of cantilevered framing. Using this view we could easily identify and change to demolition those elements that interfered with the new construction.



The last view we created displayed the final completed structure after the remodeling was finished. We set the Phase to New Construction and the Phase Filter to Show Complete, as shown here. You can't see the demolished elements. The existing model is shown in gray and the new elements are shown By Category.



The three views just described are going to be added to the construction document set to help illustrate this hard-to-understand area of the structure and its transformation to the new design specifications. You should incorporate views like these into your documents as a matter of good BIM practice.

Exercise: Working with Phases in a Project

In this exercise, you will re-model a concrete structure. One bay is being removed and replaced with steel members. You are required to create an existing view, a demolition view, and a view of the completed remodel. Your office did the original project and you have the design model of the structure that you can reuse. Your first step is to change all the current elements in the model to the existing phase since this will be an existing building now. The second task is to identify the items to be demolished. Then, the new work can be added. Finally, you will create views that display all three phases: Existing, Demo, and New.

So first let's change the properties of elements in the existing model:

1. Open the model of the concrete structure, `Dataset_CH1601_begin.rvt`, (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`).

2. Click the Default 3D View button (the house icon on the View toolbar) to access the full model view. Use the Zoom Extents tool if necessary.
3. Select the two floor decks, then click the Element Properties button on the Options bar to access the Element Properties dialog box.
4. Scroll down and change the Phase parameter for the objects to Existing. Click OK to apply and exit. Deselect the floors.
   Notice that the slabs are no longer transparent because they have assumed the Phase Existing material display. You will change that next.
5. On the menu bar, click Settings ➢ Phases ➢ Graphic Overrides.
6. Click twice in the Material field of the Existing category, which displays the Materials dialog box for Phase – Exist.
7. On the Graphics tab, in the Shading section change the Transparency value for the Phase – Exist material to 10%. Click OK twice to save and exit the dialog boxes.
8. Select any column, then on the menu bar click Edit ➢ Select All Instances.
9. Click the Element Properties button on the Options bar, scroll down, and change the Phase option for the selected columns to Existing. Click OK to apply and exit.
10. Select one spread footing and repeat Steps 8 and 9.

It would be nice if you could select all the elements at once and change their Phase setting to Existing, but that is not possible. Therefore, you need to isolate the different model object categories individually. The easiest way is to filter your selection set:

1. Select the entire model with a crossing window. On the Options bar, click the Filter Selection button.
2. Click Check None, then only check Beam Systems. Click OK to close.
3. Repeat Steps 8 and 9. That will change all the in-fill beams to the Existing phase.
4. Repeat the previous filtering steps and select all the Structural Framing (Other) elements (the girders) and repeat Steps 8 and 9.

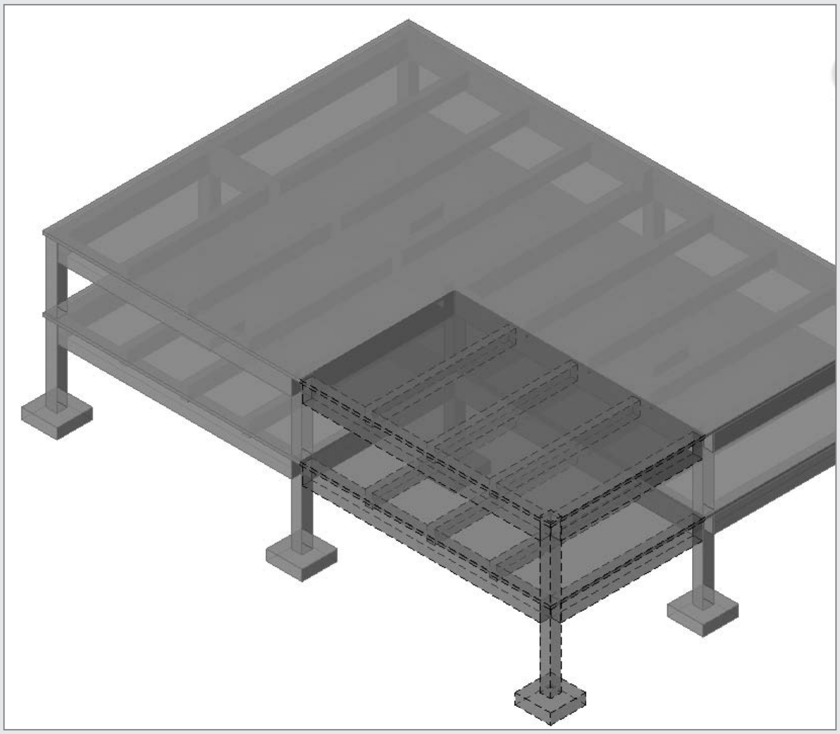
Now all of the elements have been changed to the Existing phase and you are ready to demolish objects:

1. In the Project Browser, double-click Level 2. You will demolish all the elements in the lower-right bay, from grid 2B to grid 3C. Leave the girders that run along grid 2 and along grid B.
2. Click the Demolish icon on the Tools toolbar.
3. Hover your mouse over the bottom-right bay until you highlight the beam system. Click to demolish its members.
4. Next, demolish the edge structural framing in the bay on lines C and 3.
5. Go to Level 3 and repeat the demolition process.
6. Finally go to Level 1 and demolish the column and spread footing at grid 3C.

Click the Default 3D View on the View toolbar and note that those elements are now shown in red and with hidden lines. The last item to demolish is part of the existing floor on levels 2 and 3. This process is a bit different because rather than a whole object being demolished, only part of this one will be removed. How are you going to deal with that? Well, you will create a solid void that cuts the area out. Of course you do not want the void to appear in the existing view of the structure, but by phasing your views correctly, you will be able to display all the conditions correctly.

1. Go to the Level 2 plan view. On the Modelling tab, click Create.
2. Select Floors for the Family Category and click OK.
3. Name the element **Level 2 floor demo** then click OK.
4. On the Family menu Click Void Form ➢ Void Extrusion.
5. Click Lines on the Sketch bar, then draw a rectangle of lines from the edges of the existing girders to the edges of the slab portion to be removed.
6. On the Sketch bar, click Extrusion Properties and set the Extrusion End to –**6**″.
7. Click Finish Sketch.
8. On the Tools toolbar, click the Cut Geometry button, then select the floor and the void that will cut it.
9. Click Finish Family. Perform the same process for the Level 3 floor slab.

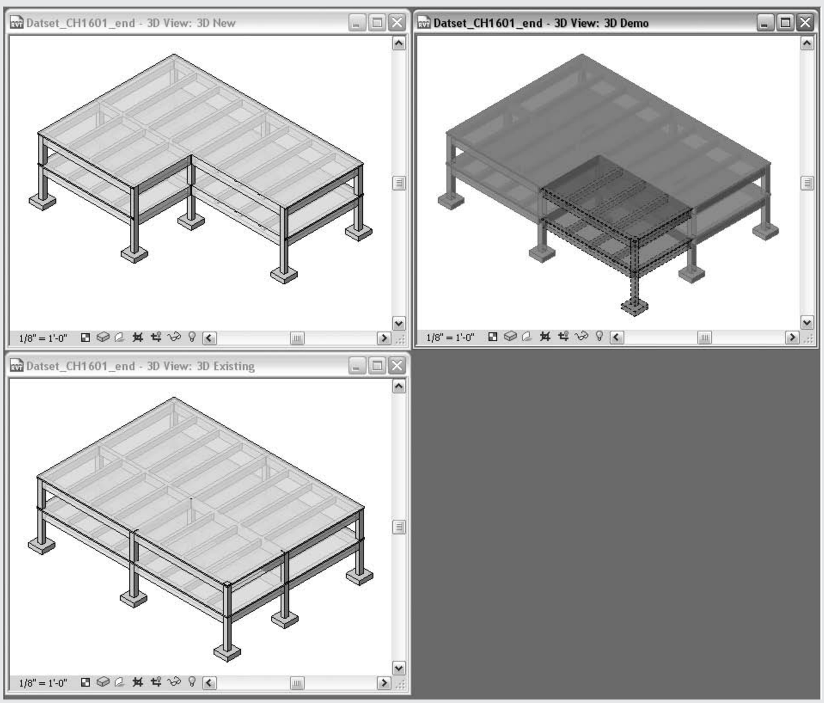
At this point, your 3D view of the model should like this:



Now you will add the New Construction to the project. First, go to Level 2. Right-click and select View Properties from the context menu. Scroll down and notice the phase settings: Show All for Phase Filter, and New Construction for Phase. Remember that the Show All phase filters are showing as existing, demo, and new construction. That works fine as a Demolition view but not as a New Construction view.

1. In the Project Browser, highlight the Level 2 plan. Right-click and select Rename from the context menu. Rename the view: Level 2 Demo. Click No when prompted to rename the level and views.
2. Right-click in the drawing area and click View Properties. Change the Phase Filter parameter to Show Previous + Demo, then click OK to exit. The phase is now set to New Construction.
3. In the Project Browser, highlight the Level 2 Demo plan, right-click, and choose Duplicate View ➢ Duplicate with Detailing. Rename the new view **Level 2 Existing**.
4. Go to the Level 2 Existing view, right-click in the drawing area, and click View Properties. Change the Phase option to Existing, then exit.
5. In the Project Browser, highlight the Level 2 Demo plan view, right-click, and choose Duplicate View ➢ Duplicate with Detailing. Rename the view **Level 2 New**.
6. Go to the Level 2 New view, right-click in the drawing area, and click View Properties. Change the Phase Filter option to Show Complete, then exit.
7. In the Project Browser, highlight the {3D} view. Rename it to **3D Demo**.
8. As you did for the floor views, create two other 3D views, one for the existing and one for the new phase, and set their phases accordingly. Name them **3D New** and **3D Existing**.

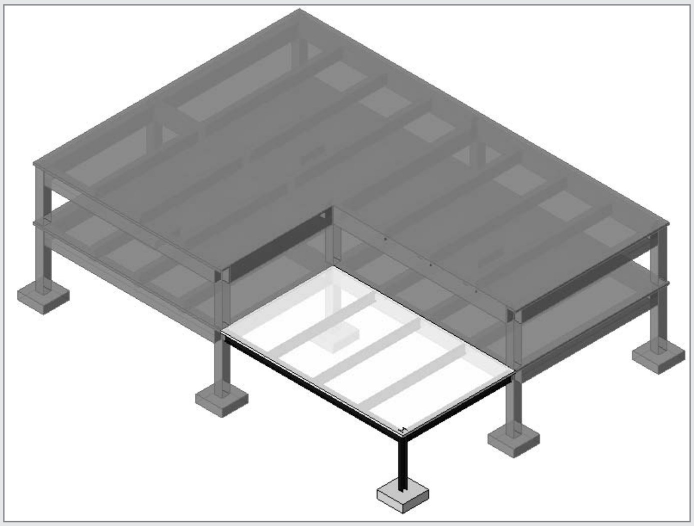
Check out the following graphic of the 3D views you just created. Is that cool or what? Of course you can create multiple views for your other floors as well, but we will no go that far in this exercise. But you are not done yet!



Now you will add the new construction to the model:

1. Go to the Level 2 New plan view.
2. On the Modelling tab, click Structural Column.
3. Add a W10×49 wide flange column to grid 3C.
4. Click Beam and use the Type Selector to assign W21×50. Then place a girder between grids 2C and 3C, and one between grids 3B and 3C. Change the girder's Start and End Level Offsetsetting to –**5″** below the floor level.
5. Click Beam System and select W18×35. In-fill the bay with beams by selecting the grid 3 girder.
6. Click Slab on the Modelling tab. Add a 3″LW Concrete on 2″Composite Metal Deck type floor into the new area. Use the lower-left and upper-right corners for a rectangle sketch box.
7. On the Modelling tab click Foundation ➢ Isolated and add a 5′-square footing for the new column. It will be placed at the bottom of the column and not visible on this view.

And there you have it. Pull up your various plan and 3D views and play around with the phasing and phase filters to see how the options display your elements. Your final completed remodel in your 3D New view should resemble the following,



If you want the existing construction to display as completely grayed out, you need to adjust the phase filter in the following way:

1. Go to the 3D New view you created earlier.
2. On the menu bar click Settings ➢ Phases.
3. Select the New Construction phase, then click the Phase Filters tab.
4. In the Show Complete line, change the Existing option from By Category to Overridden. Click OK to exit the dialog box.

Now all the existing objects in the view will be displayed in gray tone, helping you to distinguish them from the new work.

That ends our discussion on project phases, which as you have seen are very powerful as well as a bit confusing at first to understand. Next we will examine design options and how they are used in your project.

Working with Design Options

Design options are at the core of the design process. Professional architects and engineers bring life to an idea that slowly resolves itself into precise details and specifications that can actually be built. During that process, at every stage, various options are considered, analyzed, and then accepted or rejected based on their merits. Juggling multiple options and producing the graphics necessary to display each one can be a time-consuming and confusing process. Using the design options in Revit Structure can greatly ease the difficulty of managing all those design ideas. Using design options in your projects can save you considerable time and energy because you can display multiple design ideas without having to duplicate files. You can then keep them organized in one file, where each option can be displayed in a particular view.

So let's dig into the particulars and see how this process works. First you will see how to create the basic design options in your project.

How Design Options Work

Design options are created as sets of modeled elements. A building project can have multiple option sets. Figures 16.10 and 16.11 show one set containing two options for new tunnel walls at a sports stadium project. One option within the set is for a concrete tunnel wall, and the other is for a CMU tunnel wall. The new tunnel is meant to connect the locker rooms to the main playing field. More options could be added as well for the tunnel—for instance, another option for a wider tunnel. The option set was created because the contractor wanted to compare prices and materials required for the different possibilities.

FIGURE 16.10
The Concrete Wall option

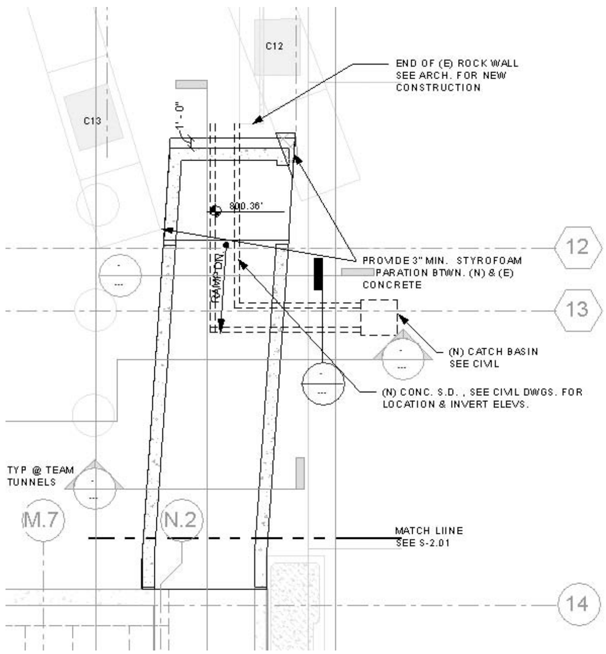
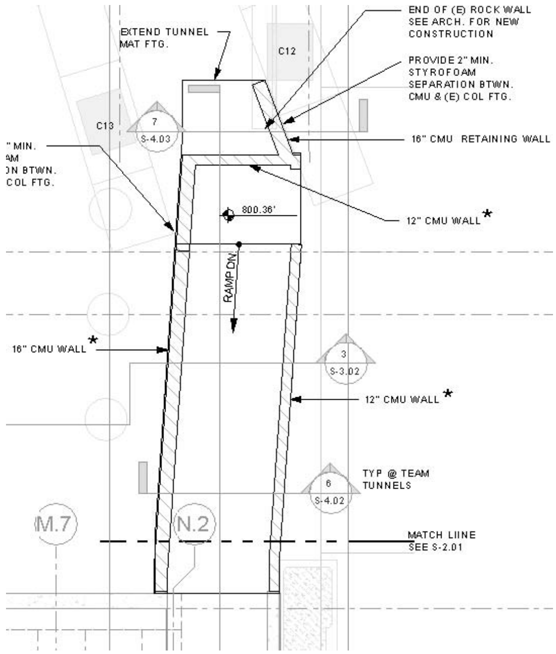


FIGURE 16.11
The CMU Wall option

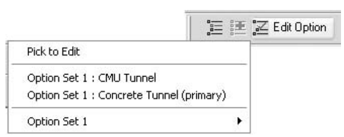


No matter how many options are added to an option set, only one can be displayed in a view at any one time. That one is called the Primary option. The others are referred to as Secondary options. Each view is then set to display one of the options in an option set. So create multiple views of an area, and in each view make one option primary for that view. This works well in the schematic and design development phase of a project as you organize your ideas and fit them to the whole design. Then as you start finalizing your design in the construction document phase, you can eliminate the views of those options that you do not want to use.

Creating Design Option Sets

Now let's look at the tools and methods you will use to create a design option set. The Design Options toolbar by default is not enabled. To enable it, hover your mouse over the toolbar area, right-click, and select Design Options from the context menu that appears. You can also access these commands by choosing Tools ➢ Design Options from the menu bar. The toolbar features three buttons: the one on the left accesses the Design Options dialog box, the middle one adds elements to a design option, and the right one enables an existing option in a view. Figure 16.12 shows the Design Options toolbar with the Edit Option button selected. Elements that are in design options are not editable until the option is enabled. You can select an option set from the dialog box, or you can click it in the view.

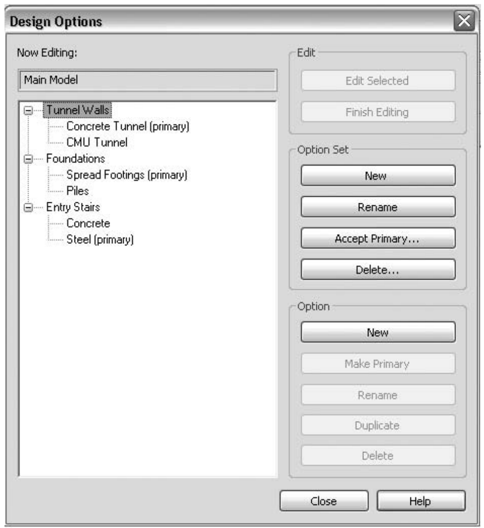
FIGURE 16.12
The Design Options toolbar



The Design Options dialog box lists each design option set, and, for each set, the option that the view is currently displaying. As you can see in Figure 16.13, the sports stadium project has two other design option sets as well. One option is for Foundations and one option is for Entry Stairs. The Foundations option set has an option for Spread Footings and one for Piles. The Entry Stairs option set has options for Concrete and Steel stair types. By default, the main model will display the primary option of the set. To change the option in a set to primary, simply highlight the option and click the Make Primary button under Option.

But what if you want to completely hide an option in a view? Maybe you have an alternative design that you don't want to show your client yet. In this case create an option in your set that has no elements and make it the primary option. When you are ready to show the new option to the client you can then Make Primary that particular option you have been working on.

FIGURE 16.13
Design option sets for our stadium project



To create a design option set:

1. Click the leftmost button on the Design Options toolbar to display the dialog box.
2. Under Option Set, click New to create the new design option set and one design option for the set.
3. Highlight the new option set, then click Rename.
4. Rename the option set.
5. Highlight the option and rename it.
6. Under Option, click New and create other options in that set as needed.
7. Create other design option sets and options in the same way as needed.

OK, that was not so bad, was it? Now that you have created and configured your design options, let's add elements to it.

Adding and Editing Design Option Elements

Now that you have created your Design Options how do you populate them with model elements? First you must enable the design option set and select the option you want to work on. Use the Edit Option tab on the Design Options toolbar to activate it, or in the Design Options dialog box, highlight an option and click the Edit Selected tab. When in editing mode if you hover your mouse over the Edit tab on the Options bar, it will display the active option.

When working in a design option, you are isolated from the main model, which becomes grayed out and whose elements are not selectable. You can override this condition by deselecting the Active Option Only check box on the Options bar, which then allows you to select elements in the main model. In a similar way, when working on the main model you are not able to select anything in the design option without first enabling it. This too can be overridden by deselecting the Exclude Design Options check box on the Options bar in the view.

All the elements that you create while in editing mode are automatically designated as belonging to that design option. In some cases, you may want to add elements from the main model to your design option. Since the main model is not selectable while you are in option editing mode, you need to add main model elements to your option before entering the option editing mode.

You do that by using the middle button of the Design Options toolbar (the one with the plus sign). Click the button, then select the main model element you want to transfer to a design option. The Add to Design Option Set dialog box appears, where you can assign the element. The element is then deleted from the main model and will only display when the option is enabled.

The following elements are not supported in design options:

Levels If you are working in a design option, you can create a level but it will nevertheless be assigned as an element to the main model. It will appear grayed out in the option.

Views Views cannot be created in design options, but rather the option is assigned to a view in the main model.

Annotations and details These elements are view specific. You are able to write text notes while editing an option, but you cannot turn them on and off if you change between options. They remain visible in the view in either case.

Some elements depend on other elements, such as a doorway in a wall. The door is hosted by the wall and cannot exist without it. The door cannot independently be assigned to a design option without the wall that hosts it. Elements of this sort include the following:

- Inserts that cut their hosts
- Host sweeps and their hosts
- Curtain panels
- Window mullions
- Grids
- Topographical surfaces and building pads

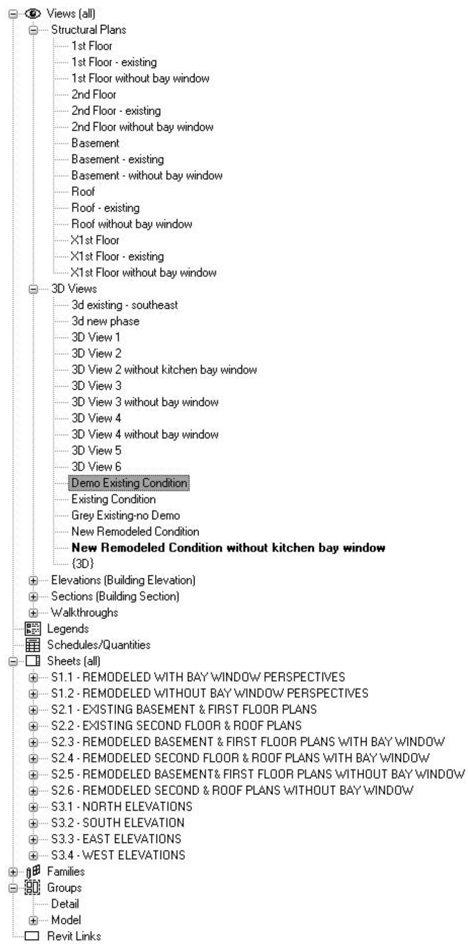
Also keep in mind that elements that are added to a group must be in the same design option as the group.

You have added elements to your design option. Next you must configure your views to display the options that you have created.

Displaying Design Options in Your Views

So far you have learned how to create design option sets and how to add elements to the options in those sets. The next piece in the puzzle is to configure your views to display the options that you created. These are often called *dedicated* views since they are developed to display only one design option. Figure 16.14 shows the Project Browser for the Williams-Drummond House historical remodel project. Study how the various views are set up. There are three basic view types that were created in the project: one for the existing structure, one for the option with bay windows, and one for the option without bay windows. Each view is configured to show a particular phase and design option. Then the various views are dragged onto the appropriate sheets.

FIGURE 16.14
The mix of views representing various phases and design options



To add a design option to a particular view, open the View Properties dialog box and select the option you want in the Visible in Option parameter (see Figure 16.15). Elements in that option will then become visible in your view. At any time you can change the view to another option by using the same parameter.

FIGURE 16.15
Adding a design option to a view

Parameter	Value
Underlay	None
Underlay Orientation	Plan
Orientation	Project North
Wall Join Display	Clean all wall joins
Discipline	Structural
Color Scheme Location	Background
Visible In Option	Option Set 1 : Wall with Bay Window (primary)
Identity Data	
View Name	1st Floor
Dependency	Independent
Title on Sheet	REMODELED FIRST FLOOR WITH BQAY WINDO
Sheet Number	S2.3
Sheet Name	REMODELED BASEMENT & FIRST FLOOR PLANS WIT
Referencing Sheet	
Referencing Detail	
Default View Template	Structural Framing Plan

You can develop a fairly complicated set of views when you also include phasing of the project along with design options, such as that in the residential project. The elements in the various design options you create can be phased to show construction sequencing. For instance, the existing structure of the historical residence in Figure 16.16 shows a 3D view of the back kitchen area and porch side of the house.

- Phase is set to Existing.
- Phase Filter is set to Show Complete.
- The design option is not relevant since only the existing elements are being displayed.

FIGURE 16.16
The existing Williams-Drummond House

Figure 16.17 is a view that shows the existing elements that remain after demolition. Along with that, the new construction is displayed with the bay window.

- Phase is set to New Construction.
- Phase Filter is set to Show Complete.
- Design Option is set to Wall with Bay Window.

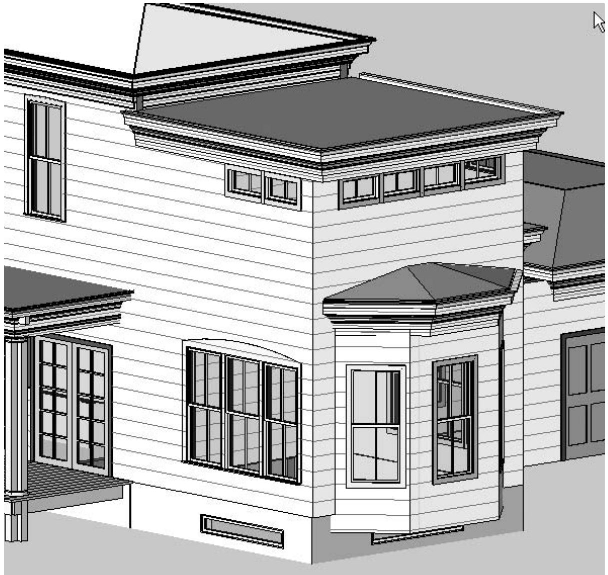
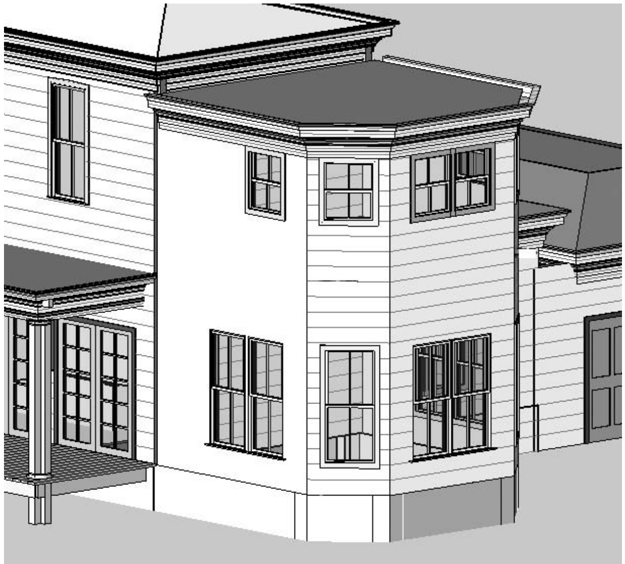


FIGURE 16.17
The remodeled Williams-Drummond House, with Design Option set to Wall with Bay Window

Figure 16.18 is a view that shows the existing elements that remain after demolition as well. Along with that, the new construction is displayed without the bay window.

- Phase is set to New Construction.
- Phase Filter is set to Show Complete.
- Design Option is set to Wall Without Bay Window.

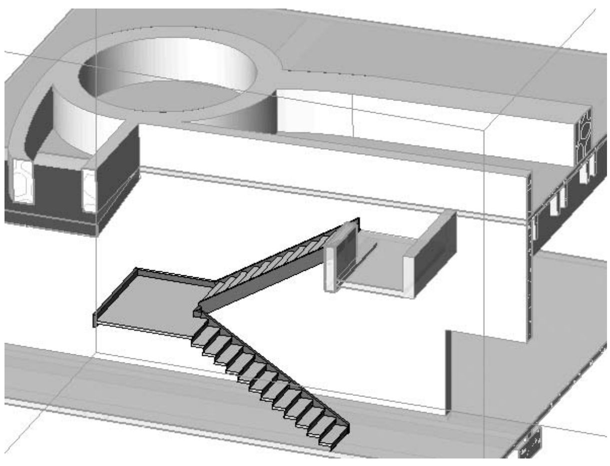
Figure 16.18
The remodeled Williams-Drummond House, with Design Option set to Wall Without Bay Window



As with phasing, the creation of design options can be confusing at first, so try something simple to build up your knowledge of the subject. In the long run, you will be glad you did. Figures 16.19 and 16.20 illustrate an easy example of design options. When designing the stairway leading to the locker rooms from the upper level, we debated whether to use steel or concrete. So we created an option set for the stairways with two options. We modeled the two stairways, one in steel and one in concrete and assigned them to their respective options. We then developed views and configured them to display one of the options. In the end, we went with the steel option and deleted the other option. It was quite easy to do.

What would have been an alternative way to deal with this situation? Well, we could have created two sets of stairways, then saved each one of them as a group. Then we could have erased one and replaced it with the other as needed. But that approach is rather clumsy and time consuming for you to have to manage. Why not let design options manage it for you?

Figure 16.19
Cutaway view of steel stair option at the sports stadium



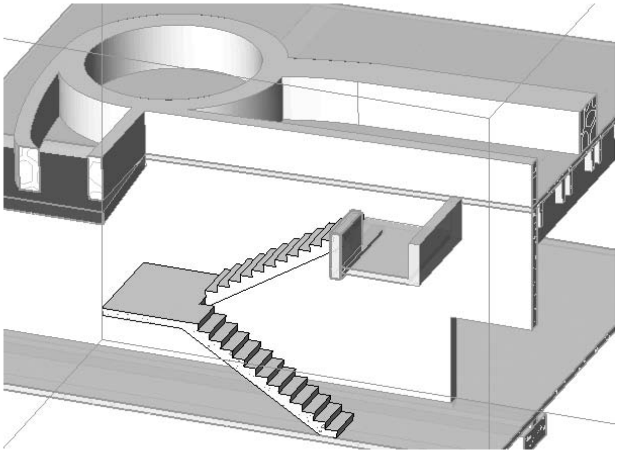


Figure 16.20
Cutaway view of concrete stair option at the sports stadium

Other Design Option Considerations

By now you should have a good grasp of the basic process for creating and using design options. Let's explore other considerations that you should take into account as you are using them in your projects:

Deleting design options You delete a design option in the Design Options dialog box. You cannot delete the primary option but only secondary options, or if there is only one option, you can delete the entire option set. Deleting will remove all elements from the model that are designated as part of that design option set.

Duplicating design options You cannot duplicate a design option set, but you can duplicate the options themselves. Simply highlight the option and click the Duplicate tab, then rename the new option.

Adding design options to the main model When you finalize your design, you may decide to incorporate your design options into the main model. You do this using the Design Options dialog box by clicking the Accept Primary button. All options will be deleted as well as all views that contain secondary options.

Designs options and wall joins Element joining can be a bit tricky to do between main model elements and elements within option sets. For instance, wall cleanup at intersections will not occur if one element is in the main model and one in a secondary option. Wall attachment to a roof or floor also will not work. In such cases, you are better off adding the main model element to your design option.

Dimensioning elements in design options You can dimension between elements in the main model and the primary design option. But secondary options work only one way: the secondary option can add a dimension to the main model. The main model, on the other hand, cannot add a dimension reference to the secondary option.

Scheduling design options The options that you create can each be scheduled separately for comparison.

So those are a few items that you need to consider as you deploy Design Options into a project. The next exercise will give you some practice in using design options.

Exercise: Working with Design Options for a Building Project

In this example you will further develop the project that you were working on earlier in this chapter. One bay was removed from an existing concrete building, and you added a one-story steel structure in its place. You also created views of the various building phases. Now your client has decided that he wants to compare the price of steel and concrete options. In this exercise you will create design option sets to document the steel and concrete alternative design schemes for the project.

First, let's create the design option set:

1. Open `Datset_CH1602_begin.rvt` and go to Level 2 New.
2. If the Design Options toolbar is not displayed, hover your mouse over the toolbar area, right-click, and select Design Options.
3. Click the leftmost button to open the Design Options dialog box.
4. Under Option Set, click New.
5. Highlight Option Set 1, then click Rename.
6. Name the option **One Story Addition**.
7. Under Option, click New.
8. Highlight the Option 1 (primary) option, then click Rename.
9. Rename the primary option to **Steel Framing**.
10. Highlight the secondary option, then click Rename.
11. Rename Option 2 to **Concrete Framing**, then click Close to exit the dialog Box.

Next, you will assign the steel elements already present to the primary option:

1. In the Project Browser, double-click on the 3D New view if it isn't active already.
2. In that view, select the two new steel edge girders, the new floor deck, the new steel column and footing, and the new beams and beam system. (A crossing window will get most, if not all, of it for you.)
3. On the Design Options toolbar, click the middle button (the one that has a plus sign on it) which will display the Add to Design Option Set dialog box.
4. Make sure both the Steel and Concrete options are checked, then click OK. The highlighted elements are assigned from the main model to each option in the set. The items are then deleted from the main model. If you try to highlight the elements now, you'll find you cannot select them until you enable one of the options.

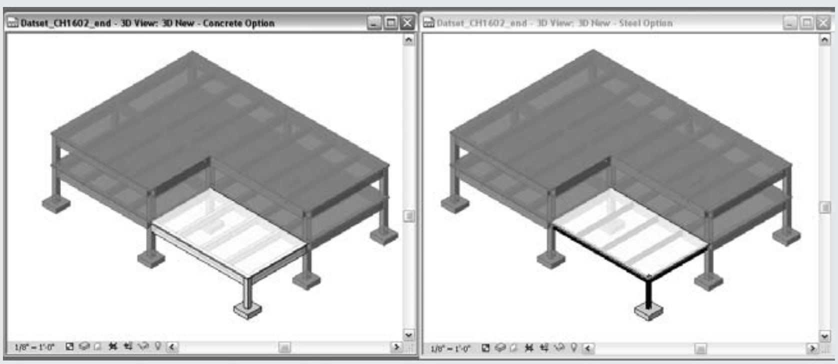
Next, you will change the steel elements in the Concrete option to concrete.

1. On the Design Options toolbar, click Edit Option and select One Story Addition: Concrete Framing to enable that option.
2. While still in the 3D New view, highlight the column at grid 3C and change it to a Concrete – Rectangular – Column: 18˝SQ.
3. Highlight the two steel girders and change them to a Concrete –Rectangular Beam: 16 × 32.
4. Highlight the beam system (you may need to use the Tab key while hovering over a beam) and change the member sizes to Concrete – Rectangular Beam: 16 × 32.

Now you will add more views to display the two options:

1. In the drawing area, right-click and select View Properties from the context menu.
2. In the Visible in Option parameter, select One Story Addition: Steel Framing (Primary), then click OK to exit.
3. In the Project Browser, highlight the 3D New view, right-click, and select Rename from the context menu.
4. Rename the view to **3D New – Steel Option**, then click OK to exit.
5. Right-click again and select Duplicate View ➢ Duplicate with Detailing.
6. Highlight that new view, right-click, and select Rename from the context menu.
7. Rename the view to **3D New – Concrete Option**, then click OK to exit.
8. In the drawing area, right-click and select View Properties from the context menu.
9. In the Visible in Option parameter, select One Story Addition: Concrete Framing, then click OK to exit.

There you have it: two options, one concrete and one steel, displayed in different views that should look like the following:



The Bottom Line

Create project phases to manage element assignments. Creating and managing phases in a project is an important task that will help establish the sequence of construction of your structure. Phases apply existing and new statuses on elements so you can manage them. A good example of using phases is in distinguishing existing elements from new when you add a wing onto a hospital complex.

Master It What steps do you take in developing phases for your building document set?

Display project phases in your project views. Views in Revit Structure are configured to display your various phases. Using the Phase and Phase Filter parameters, you set each view to display new, demo, or existing objects, or any number of construction sequence views.

Master It The phase is set to New Construction, with only the Existing phase preceding it. Describe what each of the following phase filters will show:

1. All elements from the phase before New Construction
2. Demolished elements and all of the new elements that you add to your model
3. Only the new elements that you have added to your model
4. All elements, including existing, demolished, and temporary elements
5. All elements from the previous phase that were not demolished, as well as all added new elements

Create design options to manage element assignments. In a design situation, you have to create sets of design options in order to evaluate various issues and problem areas. All of these options are created and managed in one Revit Structure file. They are then displayed in various views that you create.

Master It Answer the following questions:

1. How many options from an option set can be shown in one view?
2. What elements are not supported in design options?
3. What is a dedicated view?
4. How does using design options in your projects save you time and energy?

Display design options in your project views. Once design options are created in your project, you assign them to different views. Those views are then added to a sheet for comparison. In that way, you are able to evaluate and select primary options and discard ones you do not want as the design process progresses.

Master It Some modeled elements in your project depend on other elements and so cannot be independently assigned to a design option. Name three of those types of elements.

Chapter 17

Standards: Increasing Revit Productivity

In this chapter, you will discover how simple it is to get the "look and feel" back into your documentation. With Revit Structure you can develop templates that ensure automatic compliance with standards, while also helping you and your team focus on modeling rather than trying to develop clear and readable drawings.

In this chapter you will learn to:

- Interpret what can and cannot be done easily
- Enhance your model through customization
- Implement model standards and view overrides

Get to Work?

No doubt, the first thing you did when you or your company purchased Revit Structure was launch the software and begin making a basic model. But how did you prepare for that? Did you take a training class? Did you research graphic standard needs? Did you take your AutoCAD Standards Manual and begin mimicking that in Revit Structure? No? Exactly! Well, we have all been there. Take this new product and get it done now. So you model away for a while and then print out drawings. Then your managers take a peek and think "Yuck" when they see how poor your drawings look.

This isn't totally your fault. Most people get so excited about this new application, BIM, and 3D, that they forget about the basics. Most structural engineering firms consult for someone else, and there is an expectation of quality in the final product. But unless you carve off development time, the end product will be short in quality. Sure, the information will be there and you will have BIM and a nicely coordinated set of documents. But do these drawings convey importance? Can you tell the difference between a concrete beam and a masonry wall?

As you begin to delve into the following sections, you will expose where most of our standards come from and how to develop a plan for standards. Then you will touch on typical standard containers like fonts. We'll round off this section by examining line styles and patterns.

Know Your Limitations

When talking with other Revit Structure users, we've found that the subject of standards often crops up. Nearly everyone has some prior history with AutoCAD. If that is your history as well,

then you are probably familiar with LIN, PAT, CTB, and STB files. These Autodesk files have set the stage for standards worldwide for over 25 years. People bought AutoCAD and went right to work using the standards the software offered right out of the box.

Why did the standard from Autodesk become the *standard* for users? Simple! Because it was hard to create standards yourself and even harder to manage them. If you had the skill to develop your own custom files, you then owned the responsibility to manage them; in the AutoCAD world, those file types contain the standard. See the following list for an explanation of AutoCAD standard "containers":

- The `ACAD.LIN` file is where line patterns (dot, space, dash, and letters) are defined and then loaded into a given DWG to be assigned to objects and/or layers.
- The `ACAD.PAT` file is where hatch patterns (a regular sequence of pen strokes) are defined and then read when used within a given DWG.
- The `ACAD.CTB` file is where you define color to pen weight. The file is used only at print or plot time.
- The `ACAD.STB` file is where you define pen weight names; your DWG then references these names via layer or object overrides.

Seems like a good way to do it? It is, as long as your standards are what Autodesk suggested. But what if you don't like their standards?

- If you create new line types or hatch patterns, you then have to provide the LIN/PAT file to everyone who will need to use it for new DWG files. You can put your custom line patterns in the `ACAD.LIN` or `ACAD.PAT` or a new LIN/PAT files. You have to be sure to tell people where the LIN/PAT file is stored and to archive it with the drawings.
- If you customize a CTB/STB file, you also need to store that file on the network so that the team has access to it when they plot. Be sure to back up your files—if you lose them, you will not be able to plot properly. And make them read-only so that changes won't happen behind your back.

The joy of Revit Structure is that all four of these standard containers are in your model directly. Don't like a line type? Change it or make a new one and apply it. Want a heavy pen? Assign it where you want. Need a tighter pattern? Make it on the fly and use it right then. Not customizing Revit Structure is almost a crime—it is that easy. So don't just copy your AutoCAD standards; improve them and make them even better! And again, the great thing about Revit Structure is that your customization is contained in the model, so that anywhere your model goes, your standards go along for the ride.

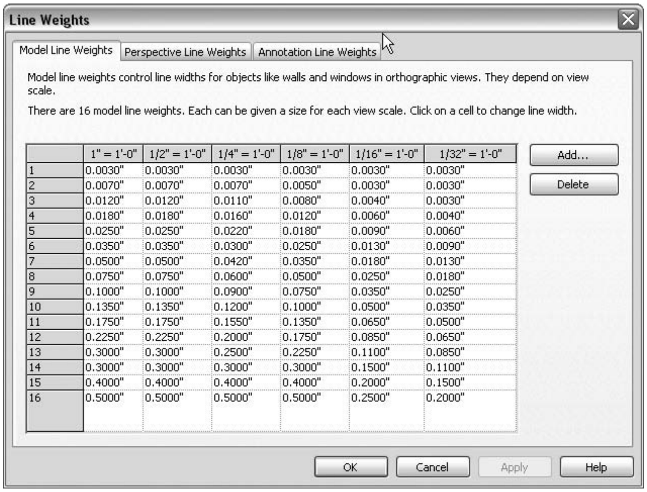
Now that you understand where historically standards have been managed and customized, it is time to learn how to begin to customize standards in Revit Structure.

Standards Planning

It is wise to have a plan of attack before you begin. When it comes to standards, you need to know where you want to go, and then build the road to get there. In Revit Structure the first step is determine the line weights you will use for your model. In the following steps you will assign these weights accordingly.

LINE WEIGHTS

Revit Structure has 16 "pen slots" you can define and use. In early versions of AutoCAD, you had 16 pen options, so to some, this will be familiar. As shown in Figure 17.1, the pens are simply numbered 1 through 16. The Model Line Weights tab lets you assign weights for model objects in orthogonal views such as plans and elevations.



	1" = 1'-0"	1/2" = 1'-0"	1/4" = 1'-0"	1/8" = 1'-0"	1/16" = 1'-0"	1/32" = 1'-0"
1	0.0030"	0.0030"	0.0030"	0.0030"	0.0030"	0.0030"
2	0.0070"	0.0070"	0.0070"	0.0050"	0.0030"	0.0030"
3	0.0120"	0.0120"	0.0110"	0.0080"	0.0040"	0.0030"
4	0.0180"	0.0180"	0.0160"	0.0120"	0.0060"	0.0040"
5	0.0250"	0.0250"	0.0220"	0.0180"	0.0090"	0.0060"
6	0.0350"	0.0350"	0.0300"	0.0250"	0.0130"	0.0090"
7	0.0500"	0.0500"	0.0420"	0.0350"	0.0180"	0.0130"
8	0.0750"	0.0750"	0.0600"	0.0500"	0.0250"	0.0180"
9	0.1000"	0.1000"	0.0900"	0.0750"	0.0350"	0.0250"
10	0.1350"	0.1350"	0.1200"	0.1000"	0.0500"	0.0350"
11	0.1750"	0.1750"	0.1550"	0.1350"	0.0650"	0.0500"
12	0.2250"	0.2250"	0.2000"	0.1750"	0.0850"	0.0650"
13	0.3000"	0.3000"	0.2500"	0.2250"	0.1100"	0.0850"
14	0.3000"	0.3000"	0.3000"	0.3000"	0.1500"	0.1100"
15	0.4000"	0.4000"	0.4000"	0.4000"	0.2000"	0.1500"
16	0.5000"	0.5000"	0.5000"	0.5000"	0.2500"	0.2000"

FIGURE 17.1 The Model Line Weights tab in the Line Weights dialog box

Now as you look at this dialog box, it may seem a little expansive with all the various scales. Don't let this deter you—all the various scales are not important and you really don't need them. The scales listed provide a means of controlling differing scales. But if your standards are like most, you will apply the same pen sizes to every scale option. An exception, however, can be made for certain types of plan work. Imagine you have a ⅛″ = 1′-0″ steel framing plan with a fairly heavy pen for the steel beams. Using a line weight scale control for 1⁄16″ = 1′-0″, you can make the heavy pen number thinner, perhaps 50% thinner, thus enabling your 1⁄16″ plan to truly look half-size.

The default setup provided in Revit Structure has a series of sizes ranging from 0.003″ to 0.5″. It is uncommon, to say the least, to see a ½″-wide line; that would eat up a lot of ink or toner. This should be your first clue that perhaps Autodesk isn't the proper source of your standards. So what should you use? What you want, of course, but here are some baseline schemes.

If you have had manual drafting training, you will recall technical pens. Those ink-filled devices for drawing crisp, black lines on Vellum or Mylar did very well back then, and their sizes can still be used today. A popular brand manufactured by Koh-I-Noor offered pens ranging from 0.13mm to 2mm wide. The goal of any line weight change is to offer a visual clue that something is to be seen more important or less important than something else. By proper assignment you can visually *layer* objects in your drawings, drawing attention to important elements and fading elements that are needed visually but don't have to stand out.

In the construction industry, the most basic drawing is a ⅛″ scale plan, so we can use that as a starting point in our setup. As an added control, you assign a size 0.025mm to pen 1. With Revit Structure, everything has a pen assignment. And with most model object defaults, Projection is set to pen 1 and Cut is often set to pen 2. By assigning pen 1 the value 0.025mm, the absolute

thinnest line a modern printer/plotter can produce with Revit Structure will be used. You will then make things thicker as required for your documentation. As shown in Table 17.1, there are many very real pens you can use within Revit Structure. Note that 0.45mm is a valid size but is not manufactured by Koh-I-Noor; however, it is available from other manufacturers.

Table 17.1 Pen Weight Table

Pen Slot	Size
1	0.025mm
2	0.13mm
3	0.18mm
4	0.25mm
5	0.30mm
6	0.35mm
7	0.45mm
8	0.50mm
9	0.60mm
10	0.70mm
11	0.80mm
12	1.00mm
13	1.20mm
14	1.40mm
15	2.00mm
16	0.025mm

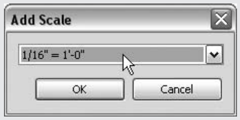
We use millimeter (mm) values here (rather than inches) because we are purists. Back in the day, technical pens used mm values, and to maintain precision, we'll do so as well. But it doesn't matter that much whether you assign 0.13mm, 0.051″, or 0.05″—only someone with a microscope and a super laser printer could tell the difference. It should be noted that once set, this value will be pretty static and should not change often. So why not be precise?

As for pen 16, it isn't needed for most work, but you could add a title such as "super thin pen" just in case it prints. You can use that pen slot as a flag of sorts for tracking changes you will make later.

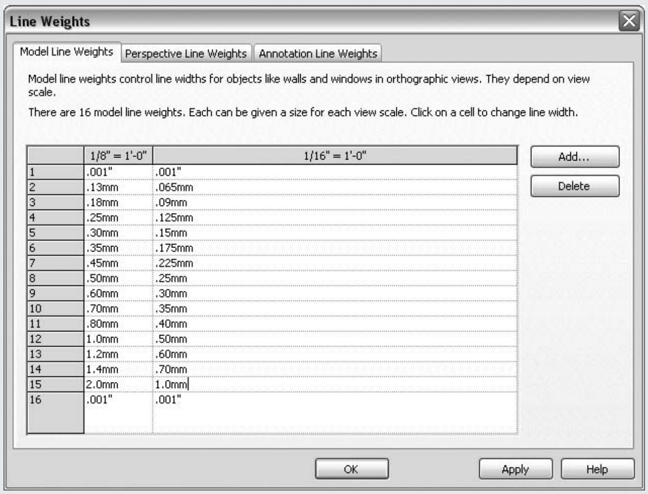
Exercise: Defining Line Weights

This exercise will examine the process for filling out the Line Weights dialog box and making adjustments to various scales. Start with a new project based on the default Imperial template:

1. Choose Settings ➤ Line Weights.
2. Select the first column on the Model Line Weights tab and then click the Delete button.
3. Repeat for all scale columns except 1/8″ = 1″-0″.
4. Fill in all sizes, as shown in Table 17.1.
5. Click the Add button and create a new scale for 1/16″. Notice how the scale adopts the sizes from the only existing column.



6. Adjust each pen width for the 1/32″ = 1′-0″ to be a half value. Just take the normal size and and divide it in half for every pen slot for 1/32″ scale.



7. Click OK to close the Line Weights dialog box.

What Pens Are Used for Undefined Scales?

If the drawing scale you're using does not have a corresponding scale in the Pen Weights dialog box, Revit Structure will use the closest, larger scale. For example, if pen 1 for ⅛″ is 0.25mm and pen 1 for 1⁄32″ is 0.13mm, if a 1⁄16″ scale plan is developed, it will use the sizes from the ⅛″ scale.

Using this knowledge, you can avoid having to assign pen weights for all scales simply so that they follow your standards. If you want all drawings to use the same pen sets, create one scale and every scale view will use it.

As shown earlier in the Line Weights dialog, you have defined two scales for your model objects. Objects shown in ⅛″ or smaller will use the ⅛″ = 1′-0″ pens, and those objects shown at 1⁄16″ or larger will use the 1⁄16″ = 1′-0″ pens. With this technique, you can have consistent quality while also getting control of conditions with small scale factors.

Line Patterns

Often referred to as a line type, Revit Structure *line patterns* are a defined series of dots, spaces, and lines that can convey a purpose. Whereas line weight in an object will indicate importance, a line pattern will help paper document readers know what they are looking at. A good example of this is a concrete beam below a slab. The lines for beams below slabs are typically drawn in a hidden line type indicating that the beam is, in fact, below.

Users of AutoCAD will access and develop new line types via a LIN file. This then will be read into an active DWG to permit use of the line type. If you need to modify an actively used line type in AutoCAD, you first have to change the LIN code (`A,.75,-.125,.125,-.125`, for example) in the LIN file, save, and then reload the file. If you create a custom pattern and then lose the LIN file, you are out of luck. There is no way to re-create the LIN from the definition stored in a DWG. It is precisely this tedious work that has forced the architecture, engineering, and construction (AEC) industry to adopt the standard put forth by Autodesk.

Where Did Revit Structure Come From?

Autodesk did not invent Revit Structure. In fact, other than purchasing the company and then spinning off Revit Structure and setting release cycles, Autodesk has done well to stay out of the way and let the people who created it keep the program moving in a positive direction.

But it is this "separation" that causes some discontinuity. In nearly all cases, you will be using AutoCAD alongside or to support Revit Structure. For example, in Revit Structure the default pattern for gridlines is Grid Line ¼″. AutoCAD, on the other hand, doesn't have gridlines. Rather, it has line types that look like a gridline (or something else defined by you) but CENTER, CENTER2, and CENTER2X do not have the same settings as Grid Line ¼″.

Revit Structure users have the option to use the standard provided in the software or develop what they want much easier than in AutoCAD. It is impossible to match AutoCAD right out-of-the-box with Revit Structure; you must customize your template to get an identical line style appearance. But do what you like and what looks good rather than simply adopting a standard that goes back 25 years.

To develop standards, look back at old hand-drawn plans, sections, and elevations to see what the *hand* would typically do. You will notice that hand-drawn lines tended to be more solid and continuous than what typically CAD systems display. Knowing why is subjective, of course, but if you think about it, it was probably done that way for two reasons. First, it took more effort to lift and drop the pen when creating long line work. But also you see that the drafter often kept the line breaks away from important intersections. For example, the line work would be continuous at column and grid intersections. The dashes would be placed between grid intersections instead. In Revit Structure, a 30′-0″ bay will have eight breaks (using Grid Line ¼″) in the line work at ⅛″ scale. Back on the board (manual drafting), this same distance would instead have no more than three breaks and would be drawn with a very thin pen.

Protect Custom Standards

You might think about redefining patterns or other default settings in Revit Structure. We recommend that you stay away from this practice. It is easy to create new definitions and just as easy to overwrite them. You could spend time redefining Grid Line ¼″ and then later you load in standards from a client or default template file. All your settings will be reset with no way to re-create them easily.

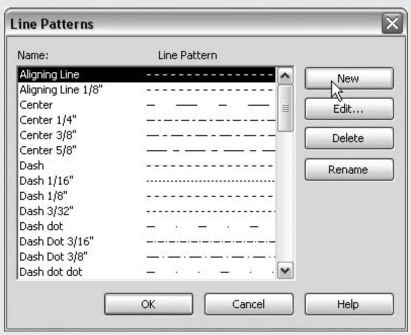
We recommend that you develop a naming protocol to provide a level of safety for your custom styles, patterns, families, and so forth. For example, you could use a # sign as a prefix in front of anything new you create. Not only is it unique (it's not used by Revit Structure), but also it will be shown first due to alphabetic listings. Use this, or another character or company initials, but you certainly should not just redefine any out-of-the-box standards.

So the first step to improve your Revit Structure standards is to define a new, longer pattern for gridlines to use and then apply it.

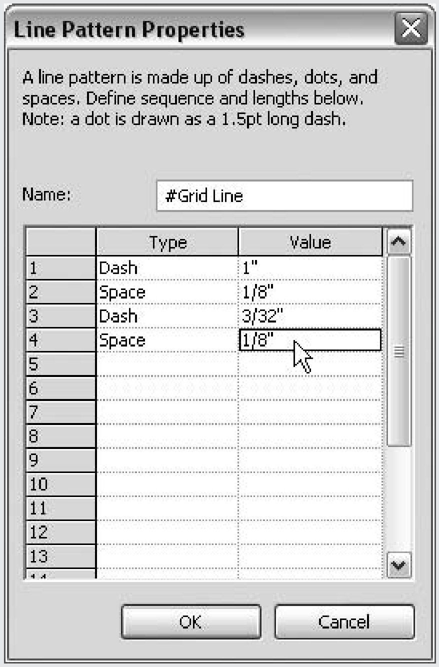
Exercise: Creating a New Line Pattern

This exercise shows the steps to create a new line pattern and then assign it to Grid Line objects. Start with a new model based on the default Imperial template:

1. Choose Settings ➢ Line Patterns to display the Line Patterns dialog box, as shown here:

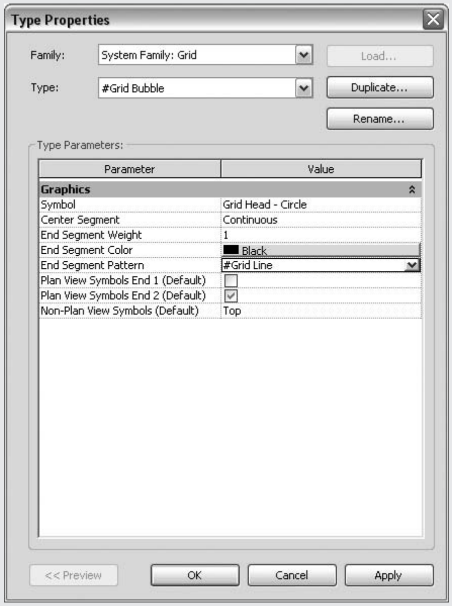


2. Click New to display the Line Pattern Properties dialog box.
3. Type **#Grid Line** as the name and the use the values shown in the following graphic for the various fields. Click OK to close the dialog box.



4. Click OK to close the Line Patterns dialog box.
5. If it's not displayed already, switch to any ⅛″ scale plan view.
6. Click Drafting ➢ Grid and create a vertical grid in the center of the drawing area.
7. Copy the grid to the right 15′-0″.
8. Select grid 2 and then click the Properties button next to the Type Selector. This will display the Element Properties dialog box.
9. Click the Edit/New button to access the grid type properties.
10. Click the Duplicate button. Name the new grid type **#Grid Bubble** and click OK.

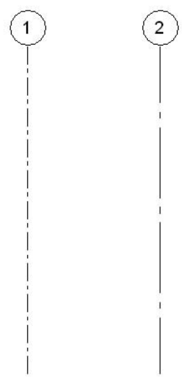
11. Once back in the Type Properties dialog box, select the End Segment Pattern, scroll to the top of the list, and choose #Grid Line. The new #Grid Bubble type is now assigned to use the new #Grid Line pattern.



12. Click OK to close the Type Properties dialog box and click OK again to close the Element Properties dialog box.

As shown in Figure 17.2, grid 2 now has a much less busy line pattern. It is less likely to be misinterpreted as something else and will be more pleasing to the eye. Also, when placed closely to a hidden line, the grid pattern will not compete with the hidden line. Again, this aids in recognition and interpretation.

FIGURE 17.2
A comparison between the out-of-the-box grid and a new custom grid type.



If you can keep in mind how your model (drawings) look and work to keep them clear, you will make your future drawing reader much happier. It takes a conscientious effort to keep those readers in mind. Remember they don't have a computer display which they can zoom in with to see something more clearly. Developing your standards is a twofold process—creating the style and then applying it.

Controlling Object Line Patterns

Revit Structure elements, like columns and beams, are managed in much different ways than grid type objects. Access to the multitudes of model objects is done via the Visibility/Graphic Overrides and Object Styles dialog boxes. If you want to set how the entire model is displayed, you use object styles. If you only want to change a single view, you use visibility/graphic overrides. In general, try to get the model itself looking right for the majority of views and then, if needed, tweak specific views.

Our first step is to learn what we have to start with. Start a new model and then make some basic objects: a set of intersecting grids, a concrete slab with supporting beams, and a column below and one being cut by the view range. As shown in Figure 17.3, this doesn't exactly pop visually.

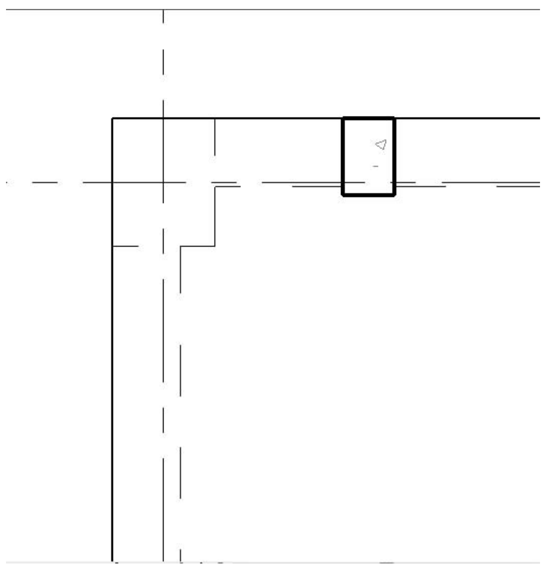


Figure 17.3
The basic objects for concrete models and their default line weights and patterns

Next, adjust your pen weights as described earlier in this chapter. The lines will begin to change weight—but not really for the better. That's because you have yet to tell the objects what weights to use!

Then you can create new line patterns for gridlines and hidden line patterns for beams and columns below the slab. Finally, you fire up the Object Styles dialog box and begin to adjust the line weight and line pattern assignments. Once complete, the grids, columns, and beams all should look different and ideally better.

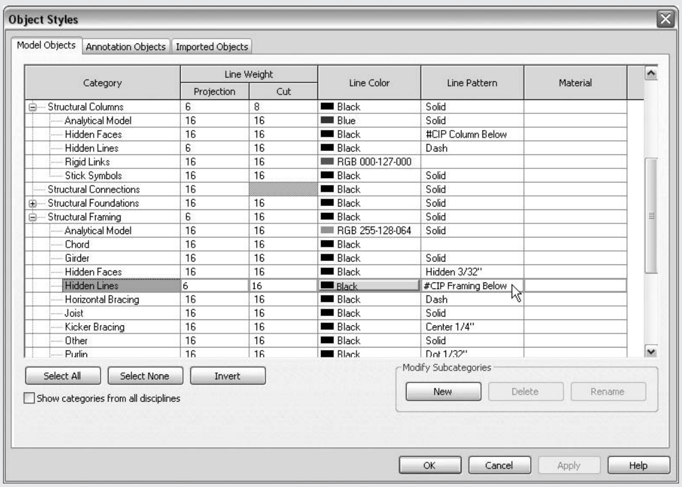
Tracking Your Changes

If you recall from earlier in this chapter, pen 16 will plot super thin and since no default Revit Structure setting uses pen 16, you can easily see what has not yet changed as you work to adjust your standard requirements.

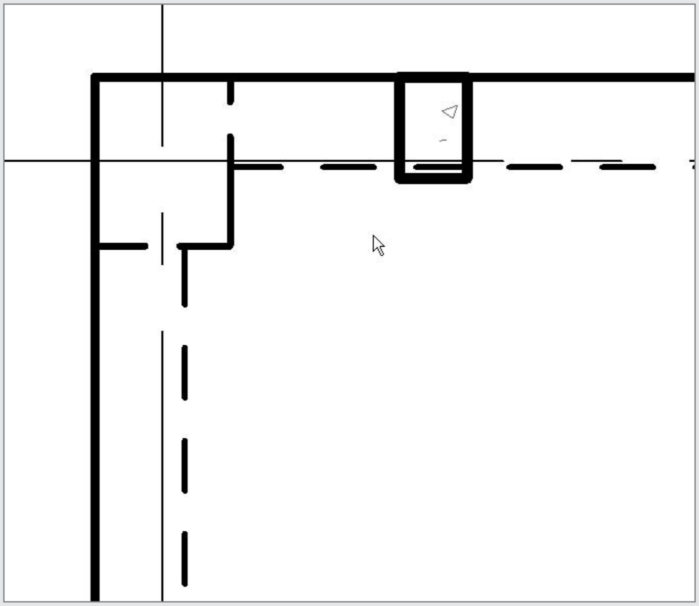
Exercise: Assigning New Line Weights and Patterns

This exercise will walk you through the steps to assign line weights and line patterns. Open `Chapter 17 Line Weights.rvt` (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`). This model has the proper line weights defined, the #Grid Line line pattern, as well as several line patterns for concrete objects.

1. Choose Settings ➢ Objects Styles to display the Object Styles dialog box.
2. On the Model Objects tab, expand by clicking the [+] icons in front of all Revit Structure categories. You will need to use the scroll bar to see and expand all categories.
3. Click the Select All button. Then use any Projection column slot and change the value to 16. Repeat for the Cut column, and change all Cut values to 16.
4. Click OK to close the Object Styles dialog box. This will collapse all the Categories. Then open the Object Styles dialog box again.
5. Select the Floors *Projection* cell and change it to pen 6 (0.35mm).
6. Select the Floors *Cut* cell and change it to pen 8 (0.50mm).
7. Expand the Structural Columns category. Now select the primary Structural Columns Projection cell and change it to pen 6 (0.35mm) and change the primary Cut cell to pen 8 (0.50mm).
8. Change the Hidden Lines Projection cell to pen 6 (0.35mm). Change the Hidden Lines Line Pattern from Hidden 3⁄32″ to #CIP Column Below.
9. Expand the Structural Framing category. Change both the primary Projection and Hidden Lines cells to pen 6. Finally, change the Hidden Lines Line Pattern cell from Dash to #CIP Framing Below.



10. Click OK to close the Object Styles dialog box. Your model should now match the following illustration.



You've made only the beginning steps to control your standards as you see fit. Every cell with pen 16 is a potential location to customize weight. Every Line Pattern cell that isn't set to Solid is a candidate for customization.

It will take some work to fully customize your modeling styles. The key to success is to tackle your model objects one at a time. Some users have gone so far as to set all pen slots to 16 and change all colors to something bold and easy to see, like hot pink. Then they proceed to reset hot pink to black, item by item, from within a model that has all model objects in it. Eventually when they don't see any hot pink model objects, they know they have addressed all projection/cut issues for the objects they placed while they were incorporating their line weight and line pattern standards.

Most of the work in Revit Structure is line work of some sort. Strokes of the pen can convey a great deal of information. But undoubtedly great linework will not do for all. Eventually you will need to provide something that can be read.

Annotation Standards

Another class of objects in Revit Structure are annotation objects. These are elements that only present themselves in one form for any views; they don't have a cut style and they tend to scale with the active view scale. Tags, level heads, and sections are all annotation objects.

Getting your annotation standards in place isn't as difficult as developing standards for model objects, so temporarily changing the line weight for all projects will reap little reward for the effort. Also, most annotation objects categories are just text, and all text in Revit Structure

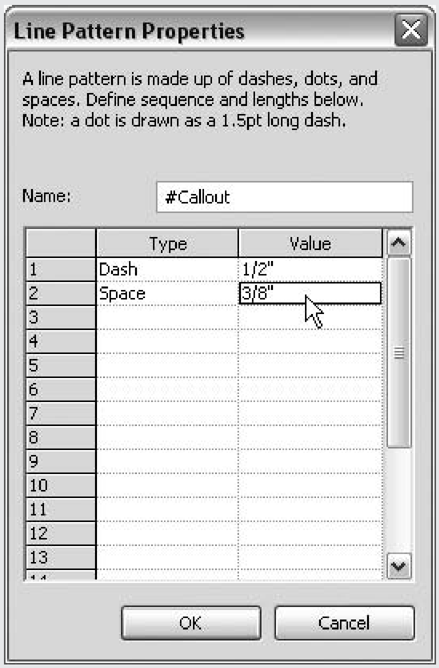
uses Windows True Type Fonts (TTF). A TTF contains the full letter stroke in it, so line weight will have little impact. Therefore, making them any pen 1 to pen 16 will be impossible to notice on-screen.

However, there are things worth customizing. For example, callouts have a boundary "box" that usually should be a medium line weight as well as unique line pattern. Also, you may prefer that grid heads have a heavier weight for the circle, and there are other examples.

Exercise: Assigning Annotation Pen Weights and Patterns

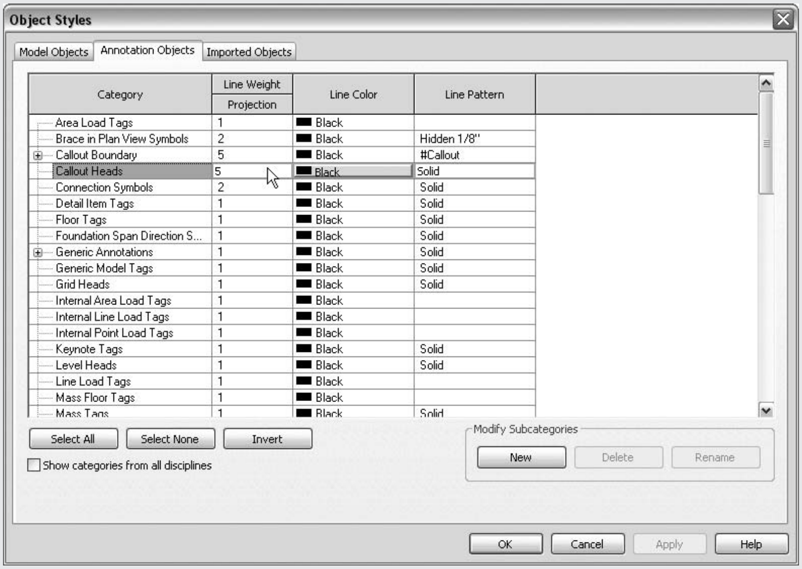
This exercise will quickly run through the steps to control pen weights and line patterns for annotation objects. Open `Chapter 17 Annotation.rvt` (from the book's companion web page). This model picks up where the previous exercise ended, but a callout has been placed around the column.

1. Choose Settings ➢ Line Patterns to display the Line Patterns dialog box.
2. Click the New button and name the new pattern **#Callout**.
3. Assign a Dash value of ½″ and then a Space value of ⅜″. Click OK to close the Line Pattern Properties dialog box.



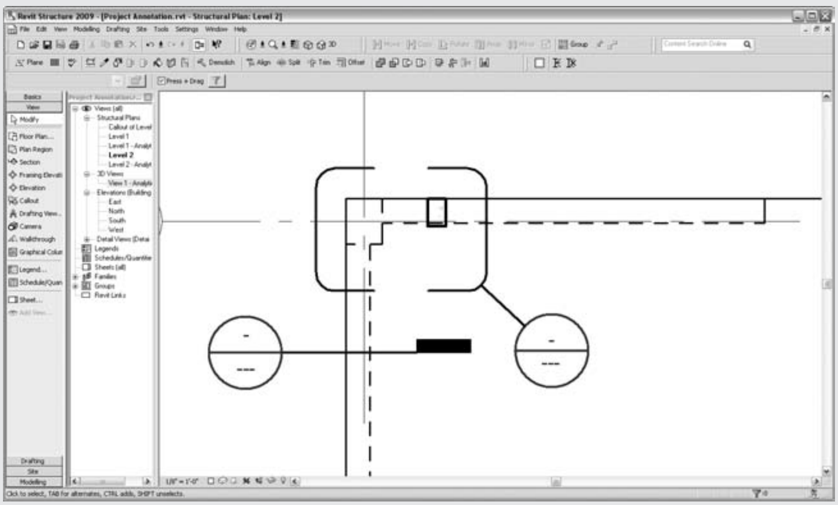
4. Click OK to close the Line Patterns dialog box.
5. Choose Settings ➢ Object Styles to display the Object Styles dialog box.
6. Select the Annotation Objects tab.
7. Expand the Callout Boundary category. Change the Projection line weight to pen 5. Change the Callout Boundary Line Pattern from Dash Dot 3/16″ to #Callout.

8. Change the Callout Leader Line to pen 5 and also change the Callout Heads category to pen 5.



9. Now change the Grid Head Projection to pen 5.

10. Scroll down to Section Line and Marks and make both slots pen 5. Click OK to close the Object Styles dialog box. As shown here, the section and callout symbols are now heavier and stand out clearly.



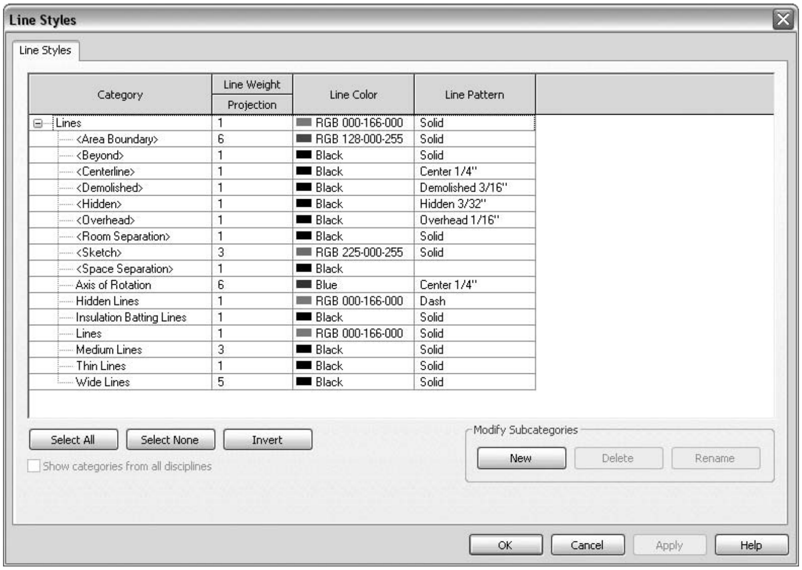
Obviously this should not be the end to changes to your standards, but you should have a better idea of the connection between the various controls at your disposal. First you define line weights, then define line patterns, then assign the pair to your object styles so that you get the desired results. This combined pair, known as a line style, is covered in the following section.

Line Styles

Once you have developed your line weights and patterns, you will probably want to use them for non-objects as well, such as drafting lines and detail components. This is done in Revit Structure through the creation and application of line styles. A line style is essentially a set of style controls saved into a single style that can then be applied to numerous objects later.

After you have defined your line styles, you will be able to use them not only to draw basic line geometry, but also to override an object's native appearance as the need arises. Creating line styles is fairly simple; a line style consists of a weight, pattern, and color. Refer to Figure 17.4 for the styles provided with the `Structural Analysis-default.RTE` file.

FIGURE 17.4
Revit Structure default line styles

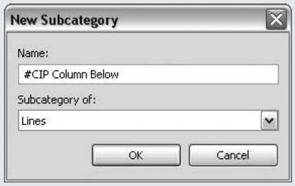


Now just as before, you would be wise to keep your standards away from default styles so that they can be protected from being overwritten, but also so that you can keep track of how your standards are organized. For example, there is a default line style named Medium Lines, and it has pen 3 as a weight. However, there is also a Generic Annotation style named Medium Lines, and it has pen 4 as a weight. These are not the same thing, so this not a problem—technically. But the possibility for confusion is high—unless your customized style is unique and you follow that naming convention across the board. A good policy is to leave everything in place and simply add your customization to a new template and then build on this later. If desired, you can purge extraneous styles as permitted by the program.

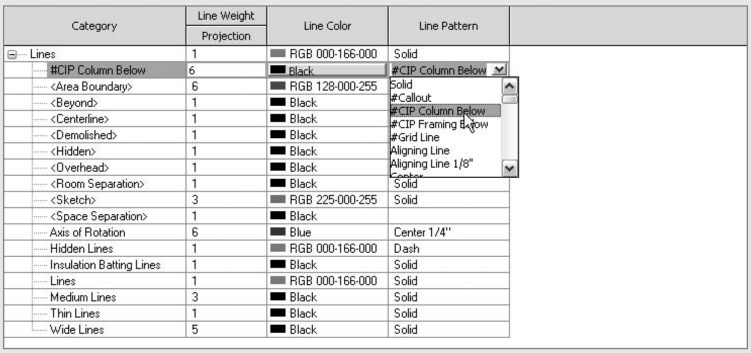
Exercise: Creating Model Object–Derived Line Styles

This exercise will step through making line styles based on model object standards. Open `Chapter 17 Line Styles.rvt` (from the book's companion web page). This model picks up where the previous exercise ended. Our goal is to create a few styles for conditions when the model must be "faked."

1. Choose Settings ➢ Line Styles to display the Line Styles dialog box.
2. At the lower right, click the New button.
3. In the New Subcategory dialog box, type **#CIP Column Below** and click OK.



4. Locate the new #CIP Column Below line style and change the Projection slot line weight to pen 6. This is the same size used in the earlier exercises for concrete column line weight for hidden line conditions.
5. Change the Line Pattern setting to #CIP Column Below and click OK to close the dialog box.



Category	Line Weight Projection	Line Color	Line Pattern
Lines	1	RGB 000-166-000	Solid
#CIP Column Below	6	Black	#CIP Column Below
<Area Boundary>	6	RGB 128-000-255	
<Beyond>	1	Black	
<Centerline>	1	Black	
<Demolished>	1	Black	
<Hidden>	1	Black	
<Overhead>	1	Black	
<Room Separation>	1	Black	Solid
<Sketch>	3	RGB 225-000-255	Solid
<Space Separation>	1	Black	
Axis of Rotation	6	Blue	Center 1/4"
Hidden Lines	1	RGB 000-166-000	Dash
Insulation Batting Lines	1	Black	Solid
Lines	1	RGB 000-166-000	Solid
Medium Lines	3	Black	Solid
Thin Lines	1	Black	Solid
Wide Lines	5	Black	Solid

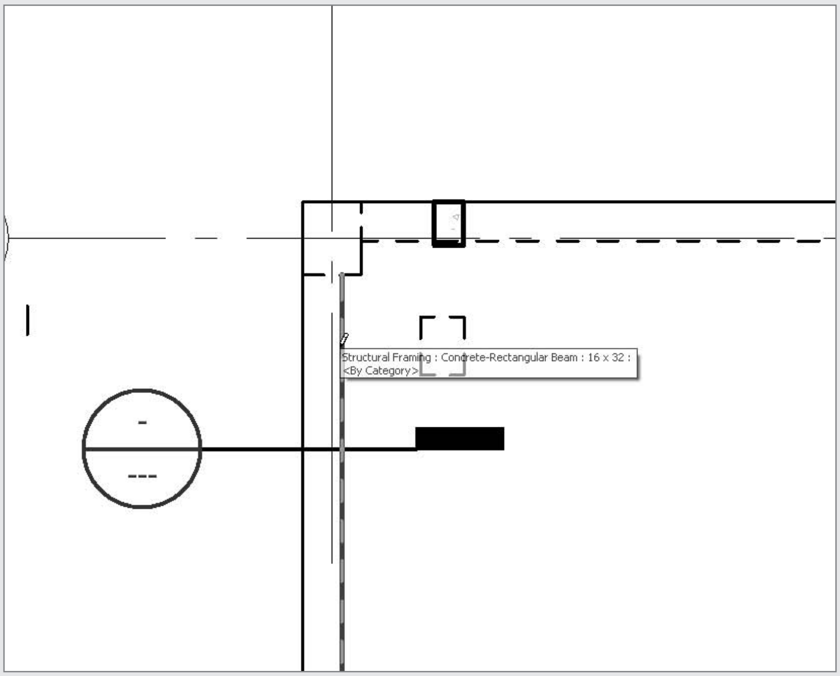
Now that you have a style, you can apply it in two primary ways. One is to draw "dumb" lines when and where required. Perhaps you just had a bunch of changes come in and there isn't enough time to fully model them.

1. On the Design bar, choose the Drafting tab and select Detail Lines.
2. Using the Type Selector, select #CIP Column Below. Choose Rectangle from the Options bar.
3. Move into the Drawing area, to the lower right of the real column below the slab. Create a box shape about the same size as the column.

Understand that this is poor modeling practice. However, it very well may save the day and you will go back later and model properly, right? The tool is there for a reason: to aid you in your work. With this, you can mimic your concrete columns below slabs. Creating other line styles that match your object style standards will also permit 2D DWG conversion into the Revit Structure "look and feel." Perhaps you are renovating an old AutoCAD-based project; with this procedure, you can import the drawing data but assign line objects for your current documentation standards for a seamless 2D/3D presentation.

Finally, perhaps you have modeled in 3D but for whatever reason the object style being presented isn't accurate. You can use your line styles to "paint" model objects. Imagine for a moment that the vertical concrete beam on the Level 2 plan view at the last minute was changed to a concrete wall below a slab:

1. Using the toolbar, locate the Linework icon and click it.
2. Change the Type Selector to #CIP Column Below and then select the right edge of the vertical concrete beam.



Notice that the line pattern changed to match the columns below. This beam now has the appearance of a column as demanded by a project change but without the hassle of remodeling just before the deadline. Again, this is not the best thing to do as a normal practice but it can be handy. If you end up needing to reset an object, use the Linework <Category> style. This will clear any *painting* you may have done. Taking the time to develop line styles that match your

modeling line work will not only provide you with a means to clean up problems but also provide a means to draw 2D just as you model in 3D.

Not everything in proper documentation is line work. Another valuable technique is to use hatching, otherwise known as fill patterns and filled regions.

Fill Patterns and Filled Regions

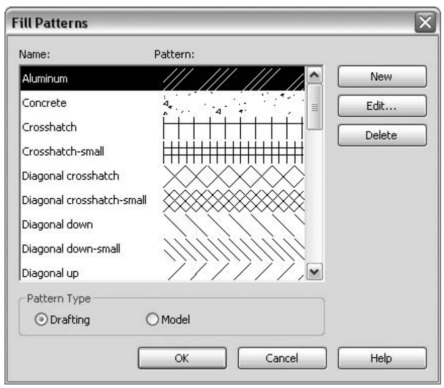
Known in the AutoCAD world as *hatch*, filled regions in Revit Structure enable you to provide tone to your drawings. It is this tone that conveys that a simple box is made of steel, concrete, precast, masonry, or some other material. AutoCAD has had a healthy selection of hatch patterns for years, and Revit Structure can leverage them—and more. Not only can you bring in your favorite AutoCAD hatch patterns, but you can also set up patterns for your model that can be used over and over again without keeping track of scale and rotation. In AutoCAD you must determine the scale of the pattern, its rotation, and layer, each and every time you place a hatch pattern on your drawings. In Revit Structure, those questions are answered when the fill pattern is defined; you decide the scale, rotation, line weight, opaqueness, and color (if needed).

An added perk with Revit Structure filled regions is the inclusion of the boundary in the filled region itself, along with a line style control for the boundary line work. And since filled regions have a type property, also known as a style, if for some reason you want to globally change a given filled region pattern you can do so and the entire model file will update with the changes.

You can place filled regions in any model view except 3D and you can place them into legends. A great use is for plan notes and legends on sheets. You also have patterns that are also printed 1:1, known as drafting fill patterns. You also have model fill patterns, which are used to represent real model objects such as tile and masonry stacking.

To define a new or adjust an existing pattern, choose Setting ➢ Filled Patterns. This displays the Fill Patterns dialog box, shown in Figure 17.5.

FIGURE 17.5
Use the Fill Patterns dialog to define your project hatch styles.



Let's say, for example, you want a honeycomb style hatch you used before in AutoCAD. It doesn't exist natively in Revit Structure, but you still have your `ACAD.PAT` file available. Open the Fill Patterns dialog box and click the New button. This will display the New Pattern dialog box. Here you can create a basic line-based pattern or, as in this case, read an existing PAT file for a pattern. At this point, click the Custom radio button. The dialog box changes, allowing you to browse to a PAT file (see Figure 17.6).

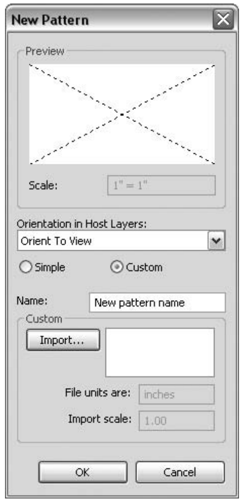


FIGURE 17.6
Browsing to a PAT file

Now, click the Import button to open the Import Fill Pattern dialog, where you can hunt down your AutoCAD-installed `ACAD.PAT` file. Normally stored in the `..AutoCAD\support` directory—find yours by typing the following at the AutoCAD command line:

```
findfile "acad.pat"
```

If AutoCAD can locate the file, it will return the directory location. After you have found your PAT file, Revit Structure will list all included patterns in the Custom list panel. Next, assign the name **Styrofoam** and then decide if you want the same 1:1 scale as in AutoCAD. In this case, the hatch scale in AutoCAD is set to one-half the old AutoCAD scale value. Revit Structure doesn't have a DIMSCALE or a scaling problem, so here you can use 0.5 to make the pattern half the size as the 1:1 scale in AutoCAD. Finally, if this pattern was going to be used for something like a wall, then you would adjust the Orientation in Host Layers setting to Align with Element. In Revit Structure your patterns can rotate with any object that uses a pattern. At this stage you are done, as shown in Figure 17.7.

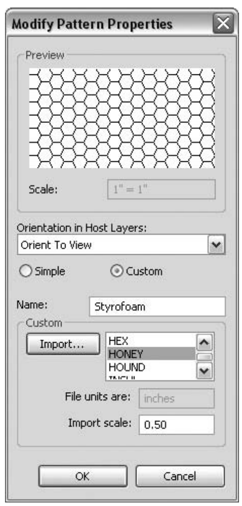
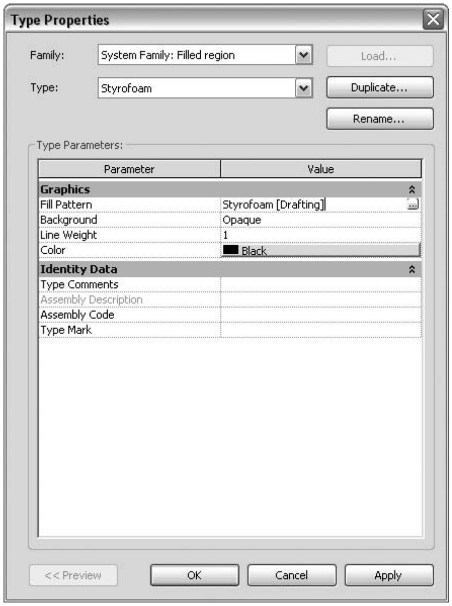


FIGURE 17.7
You've created the Styrofoam fill pattern.

Now that your pattern for Styrofoam is created, you can add it to a new 1:1 Legend view. Once within the view, use the Design bar and select the Drafting tab. At the bottom is the Filled Region button. Click this button to redisplay the Design bar as the Sketch tab. Zoom in a bit and use the rectangle option to create a 1 × 1 box. Then, click the Region Properties button in the Sketch tab, displaying the Element Properties dialog box for filled regions. Since you still need to create your filled region style, click the Edit/New button; then in the Type Properties dialog box, click the Duplicate button and set the name to **Styrofoam**. Click OK. The dialog box closes and changes the active type appropriately. Next, change the fill pattern to match. You can either use the ellipsis ([…]) button to browse for the fill pattern, or, since you know the name, you can type **Styrofoam [Drafting]** and Revit Structure will load it. Then decide whether you want your Styrofoam pattern to always hide other content by using an opaque background, or let it overlay with other line work (choose Transparent). If you never want any line work to show through the region, leave this setting as Opaque (see Figure 17.8).

FIGURE 17.8
The Styrofoam filled region type properties



Once you have finished adjusting the filled region type properties, click OK a few times to close all dialog boxes. Back in the drawing area, your rectangle is already done (see Figure 17.9). If warranted, you could change the rectangle line work by selecting the lines and then choosing another type from the Type Selector. In this case you may want the Thin Lines border. Simply click the Finish Sketch button in the Sketch tab. Then instantly your box is filled with the honeycomb pattern you defined earlier, and it is automatically a tighter pattern, matching your standards.

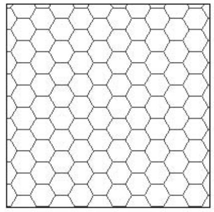


Figure 17.9
A Styrofoam hatch made using a filled region.

Adjusting Content

After you have defined your line weights, line styles, and filled regions, you can then use them on your typical details or other types of families, such as annotation symbols. You have put all this work into your standards, and these features will be applied to any new content in your model. But unless you inject them into your various family files, those nonstandard elements will propagate into your model as you bring them in.

In some cases, depending on the file type, you can use File ➢ Transfer Project Standards from your standards model into your family file. Another option is to copy/paste from one into another. Once your family file has the styles within it, you can select existing objects such as lines and use the Type Selector to switch them to your styles. Once finished, use the Purge Unused command to rid the file of the now unused styles.

This can be a tedious task, but it is necessary if you are to create and use a truly seamless single set of standards. Keep records of the files you change, or use a separate file structure for your customized files as new builds and new versions of Revit Structure families come out often and you don't want to overwrite your work accidentally.

User Customization

Once your project models can create drawings that look good, then it is your turn. Make Revit Structure easier to use! Although not inherently a visual that you can show off to your superiors, a few deft moves of your fingers on a couple of mission-critical Revit Structure support files can free up tons of time later. Hopefully this free time then can be spent on more customization or learning a seldom used feature of the program.

In this section we will explore enhancements that can be utilized to improve Revit Structure and your use of it. There is an .ini file where you can tweak Revit Structure to make it more user friendly. You can also empower Revit Structure by adding access to your own custom family files, but do it smart. Also, you can customize command shortcuts to speed your work with the software.

The Revit .ini File

The Revit `.ini` file is where Revit presets library folders, obtained during installation or a network installation image. This file contains mostly code that you are better off adjusting via Settings ➢ Options. However, there are a few gems you can only have by manually editing the `Revit.ini` file.

First determine where your .ini file is stored, which is no easy task in today's myriad of operating systems. The simplest method is to check out the properties of any Revit Structure 2009 shortcut link on your desktop or in your Start menu. The .ini file will usually be in the same location as the `Revit.EXE` file.

Once you find the .ini file, we recommend that you make a backup copy. Then open it for editing with Notepad. Now you are ready for the gems!

Temporary Dimension Values

No doubt you have noticed how Revit Structure will create many temporary and almost unreadable dimensions as you draw or edit objects. The problem is that this text is somewhat small (8 point) and can be placed poorly by the software. If you find yourself struggling to read these items, then take advantage of a tweak to make the text bigger on screen. Edit your .ini file and look for a `[Graphics]` section. If it doesn't exist, then create it at the bottom of all the content. On the line following `[Graphics]`, type **`TempDimFontSizeInPoints=`*`XX`*** where ***`XX`*** is a value normally larger than 8. A good start point is 14, 15, or 16. As you can see in Figure 17.10, the little helpful text is quite a bit bigger now.

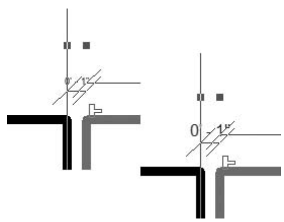


Figure 17.10
New, easier-to-see temporary dimensions

Split with a Gap

Want a free command? For whatever reason, Revit Structure has this option off by default. On the Tools toolbar is a Split Walls/Lines command. This command will break a wall or line into two segments. Pretty handy indeed, but typically with walls not only do you want to break it into two but you also want a gap. But the problem is that the newly broken wall(s) are joined at the end. If you simply grip one and drag it shorter, the other will lengthen and stay stuck on the end.

However, with the addition of a `[Construction]` section in your .ini file followed by `AllowSplitwithGap=1`, you will gain a new command on the Tools menu, right under the other Split command (see Figure 17.11).

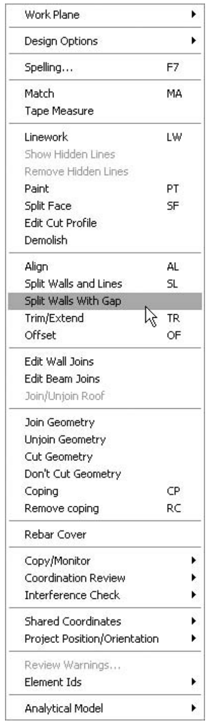


FIGURE 17.11
Instantly, a new command for breaking walls with a gap value

PRESS AND DRAG

One final helpful .ini edit is a control to set the default status of the Press and Drag feature in Revit Structure. This selection method allows you to select a group of objects with no command active and then, by clicking a point in the Drawing area, move them in whole to a new location.

If this seems like a non-issue, then you are not a new user to Revit Structure! More often than not, new users will inadvertently grab objects and move them and in some cases fail to notice. By turning this feature off in the Options bar, as shown in Figure 17.12, you can protect your model from these sorts of mistakes.

FIGURE 17.12
Turn off Press + Drag for new users.

So why the .ini file edit? Revit Structure by default has the feature set to ON or checked. Do you think you can tell your new users one time and they will remember to shut this off every time they launch Revit Structure? Obviously not, so edit their .ini file and stick this code into their file:

```
[Selection]
AllowPressAndDrag=0
```

The next time Revit Structure fires up, the Press + Drag toggle will be off, thus protecting your user from making editing mistakes. Once users learn the ropes and come back asking why they have to turn this control on every day, then you can enlighten them and remove the entry or set it to 1.

Library Paths

You are probably aware of this already, but customizing library paths not only helps to implement standards but also, with proper organization, aids in your management activities. The key is understanding why it is important, and then getting consensus on how to organize. Nothing is more frustrating than not being able to find a family you need in order to keep modeling. As you can see in Figure 17.13, the default installation will place all of your library and related family files on your local drive.

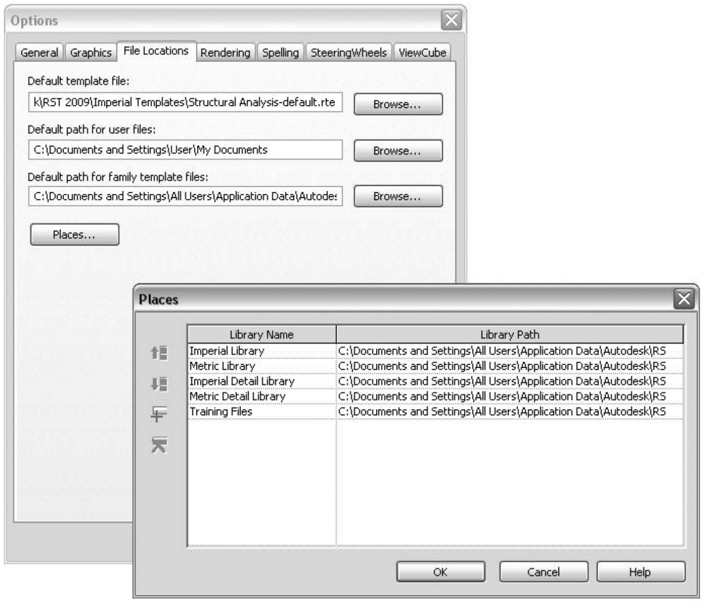


FIGURE 17.13
The default installation paths for library files found on the File Locations tab of the Options dialog box and the related Places dialog box

The first step is to develop a plan to share your library files across your network. Typically this involves assigning a directory location on your local server where each and every user has the same drive letters pointed to the same server share. For example, `\\server\software` could be then mapped to each network user as the S: drive. Then on the server you would place the Revit Structure library files as they exist normally on the local C: drive.

So at this point you have `S:\Revit Structure 2009\Imperial`; you then would have all your fellow Revit Structure users modify their Places dialog box to match the new server locations. Once this process is complete, you and your entire team will all share the same set of templates, family files, and have a new home in which to put custom files.

Library Maintenance

All is good in your world, running smoothly, and then after a little time passes you get word about a new build of Revit Structure. As a new and growing industry, Autodesk and the Revit Structure development team is always working on the next release of the program. But sometimes a bug is found, or a fix made, and Autodesk issues a patch to the software. At the same time, they may or may not include new and/or revised family files.

So you install the new build or version, and then go to copy the new library files over your existing server located files. But you pause and wonder, what changed? Anything? Everything? What if the name of your own custom family file is now used by Autodesk for a similar family they created? If you copy it over, it will wipe out yours. Did you edit the wide flange catalog file to change the X to an x? That will be replaced as well.

Did you write a diary of all the changes you made 3–4 months ago when you rolled out the then-new release? Move with caution unless you like starting over. Okay then, what can be done to avert disaster in the future?

Creating Your Directories

The recommended method is to create your own library structure and from within each directory create a shortcut to the default files. We also recommend pointing all users to your library structure within their Places dialog box. If you take the time to customize the Steel Column family, then you probably want people using it before the out-of-the-box families.

For your family files, you assign a unique naming convention to all custom or revised family files. As shown earlier, we used a # prefix for all files created or changed.

If you are modeling and select a family-based object in the model without a # sign prefix, then you know you didn't modify it. If by chance you see the same family name with a # sign prefix, then you *know* you need to use that one instead. As an added benefit, family files prefixed with a # sign will be listed at the top of their section in the Project Browser and Type Selector.

When you name all family library directories with a # sign prefix, that tells all users that this directory is the company default structure. All family files and directories are stored on the server already, so it is easy to share.

The out-of-the-box content is not under the default directories users place in their Options File Search paths. Instead, placing shortcut links in your directories, such as `..\#Column\#Steel`, would help isolate and organize any customized RFA files and a shortcut prefixed with a "`!`" to the normal out-of-the-box location `..\Column\Steel`. As a user, when you go to load a steel column family you would see #Family folders and a link to the default file location. You can then decide if you need the company standard or want the original files, in which case a single directory click will get you there

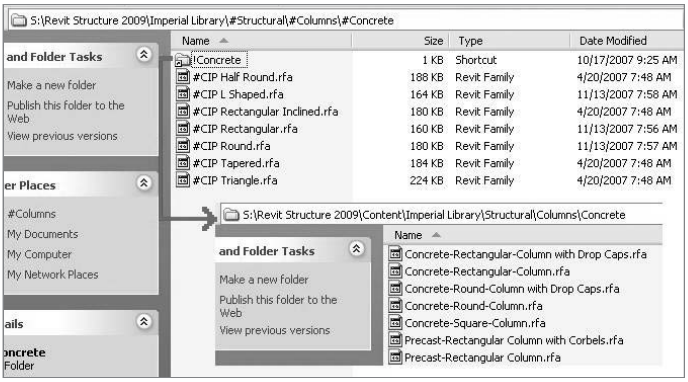
Why the !?

Why place an exclamation point (`!`) in the shortcut names? The answer is so that users can distinguish between a true directory folder and a folder shortcut. In some cases a shortcut name will read the same as a real directory. You can use whatever character or naming convention you wish, but choose something unique. Additionally, a symbol also conveys "default" or "out-of-the-box" to your users. When they click something with a "`!`" in the folder name they know where on the server they are going.

For example, say you need to load a detail component. You fire up the Load dialog box, where you are presented with all #-prefixed family files. More than likely, that is what you want, but on occasion you have not customized your own. So in that directory will be a shortcut link to the same family location under the Content folder. The user can then click the link and will be sent into the out-of-the-box location.

This probably seems confusing, so let's break it down again. There are two directory structures, A & B. A has the same organization but only contains Custom (#) family files. B, on the other hand, is right out-of-the-box. Within the A structure are shortcut links to the same directory structure location in B. The user's options are all pointed to A directory structures. Take a look at Figure 17.14 for an example of this in use.

FIGURE 17.14
A custom # directory structure and default content locations



Shortcut Keys

If you have used AutoCAD more than casually, you are probably aware of the `ACAD.PGP` file. You can use this file to define keystroke(s) to invoke a command. For example, you can use this file to specify that rather than using the menu or toolbar button, you simply press L and then Enter to get the Line command.

Revit Structure has similar customization features, albeit in the BIM world. You'll find the file `KeyboardShortcuts.txt` alongside the `Revit.EXE` file, or in your user account directory structure. Once you find the file, simply edit it with Notepad. Here's an example entry:

```
"MA"      menu:"Tools-Match"
```

The `"MA"` indicates the series of keys to be pressed. That's followed by the exact menu bar path to the command required (in this example, Tools ➢ Match). If you are not in an active command and you press M, the Revit status bar will show which command will activate when you press Enter or the spacebar. If the command shown on the status bar is not what you want, you can press any arrow direction key (normally to the right of the Enter key) to cascade through other shortcut key definitions.

Once you open your shortcut file, you will probably notice that most of the commands are not defined. But before you dive in and make a shortcut for each and every menu command, ask

yourself, what do I need? For each menu item listed, many will be used seldom, if ever. Do you really need a shortcut to something you don't use and therefore won't remember when you actually do need it? You will probably just go up into the menu bar and find it and then run from there. Commands you use every day are the ones you *should* customize.

To customize a shortcut item, find the command you want and then remove the `;` sign and type in your one- or two-letter shortcut. For example:

```
; ""      menu:"Tools-Tape Measure"
```

would become

```
"TM"      menu:"Tools-Tape Measure"
```

So then in Revit Structure, if you press TM, the Tape Measure command will start as if you chose it from the menu bar. Nearly every key on the keyboard can be used for a shortcut, even function keys (although they are little different). Since a function key has a letter and number, they are not quoted. For example, to press F6 and get the Options dialog would be:

```
F6 menu:"Settings-Options"
```

Finally, you can use the same set of keystrokes for several commands. The first one listed in the `KeyboardShortcuts.TXT` file will be run first, but you have the arrow control options to select any of the following commands with the same definition. If the order in the TXT file doesn't fit your needs, you can reorder the command so that it does. But rather than moving items to the top, place a semicolon in front of the item being relocated (to shut it off). And then place a working version at the top. This will keep your shortcut file pristine, thus enabling you to make easy upgrades.

Shortcut Planning

You should think about how you use the graphical user interface before assigning shortcuts. Normally, you will assign shortcuts to first-priority commands, such as Align and Copy, in either close proximity or using double letters. For example, Align would be AA, Copy would be CC, and Properties PP. Any toolbar icon used daily should get a shortcut that you can type with one hand easily. Next you assign shortcuts to second-priority menu item commands that you use often. Use an intelligent and consistent method so that you can remember the keystrokes—such as SY for Symbol. Those keys are easy to reach with one hand and not so hard on the memory. Now feel free to change everything if you want. Out of the box the shortcut for Structural Wall is WA. But there is also a Wall (nonstructural) command. It makes more sense to me to make Structural Wall SW instead.

As you can see, you can do a lot to speed up your work with simple and easy edits to your shortcuts. Although you cannot share a shortcut file on a network, your office should share the customization that you develop, if for no other reason to let you sit at other stations from time to time to help users understand something and not become frustrated with different command shortcuts.

Modifying Your Model

Now that you have spent considerable time figuring out what your pen weights and line styles should be, you then will want to apply them to a model file. You cannot just define these items; you must tell Revit Structure how to use them effectively for your benefit. You will make wholesale assignments and change everything in your model. Then, as needed, make changes at the view level.

Now you have the power to put Revit Structure work for you. You have modified your .ini file, and you have temporary dimension text you can actually read. You have your own custom files available to load before the out-of-the-box files. And you have managed to speed up your user interface by assigning shortcuts to your frequently used commands. After all this, you are ready to see real results in your model by adjusting your object styles and applying visibility/graphic overrides when required. Then we'll check out the concept of color and see what it can do for you.

Object Styles

One of the first things people notice about Revit Structure is that it doesn't have layers. Almost every other CAD software package has layers—how can Revit not have layers? Well, in a way it does, but not in the traditional sense. Everything in Revit Structure is an object of some sort. It could be a model object like a column, an annotation object like text, or even imported objects from an AutoCAD drawing file.

You use the Object Styles dialog box to control the model-wide look of everything in Revit Structure. To begin, choose Settings ➢ Object Styles to open the Object Styles dialog box shown in Figure 17.15. This dialog box has three tabs. The first is Model Objects, where you define how all the various 3D model objects in Revit Structure look.

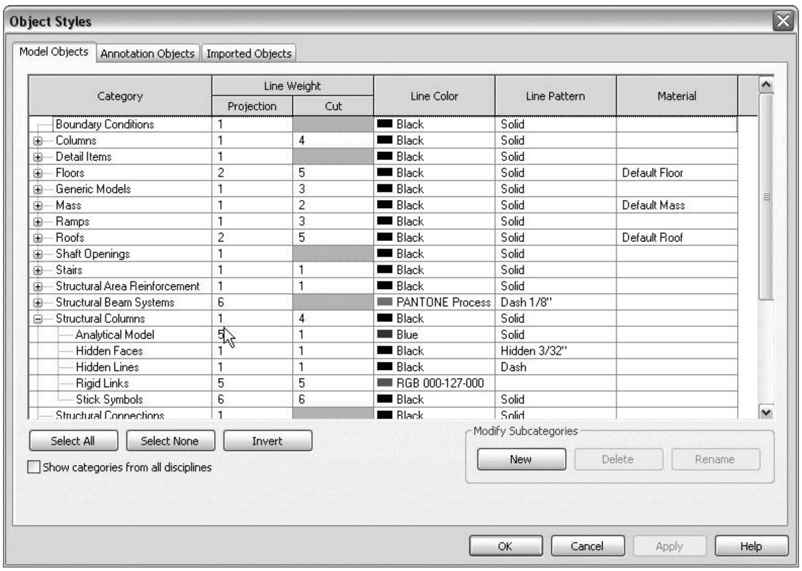


FIGURE 17.15
The Object Styles dialog box controls how all objects in Revit Structure appear.

For each object type you have a control for projection lines and, if applicable, cut lines. You also have controls for color, pattern, and, if applicable, material. Each object category will expand into applicable conditions. For example, structural columns have projection at the base level and then controls for analytical, hidden faces, hidden lines, rigid links, and stick symbols.

Learning Object Styles

It is beyond the scope of this book to explain each and every category. But there is a trick I used to learn and understand the controls. Take a model with all your various model types used and change all object styles controls to some very obvious color of your choice. Now look at each object type one at a time, and one by one change each row in the Object Styles dialog box back to black (or another unique color). Look at each object type in plan, section, and elevation views and in each detail level (fine, medium, coarse). As you change a given object style's row to another color, look at your views and note what is changing. You will soon learn that Floors – Slab Edges represents edge angles in plan views and that Floors – Common Edges apply to the line work between metal deck and concrete slab in medium/fine section views when the deck is running parallel with the view. Before you deal with the next object type, set the current one you just examined (and now understand) and change it back to black. This will help you avoid backtracking as you work your way through all the object types.

We also suggest that you set rows in which you don't see any change to an odd color, like hot pink. That way, you'll know in a glance that either you don't use that object type or you simply haven't been able to create a view to make that given property show itself. Do this in a real project and tell all team members to contact you if they ever see anything "hot pink" in color. You suddenly have expanded your investigation team beyond yourself, and before long your additional eyes will probably see your mysterious condition. Then deal with it, understand what it controls, and assign proper line weight and pattern.

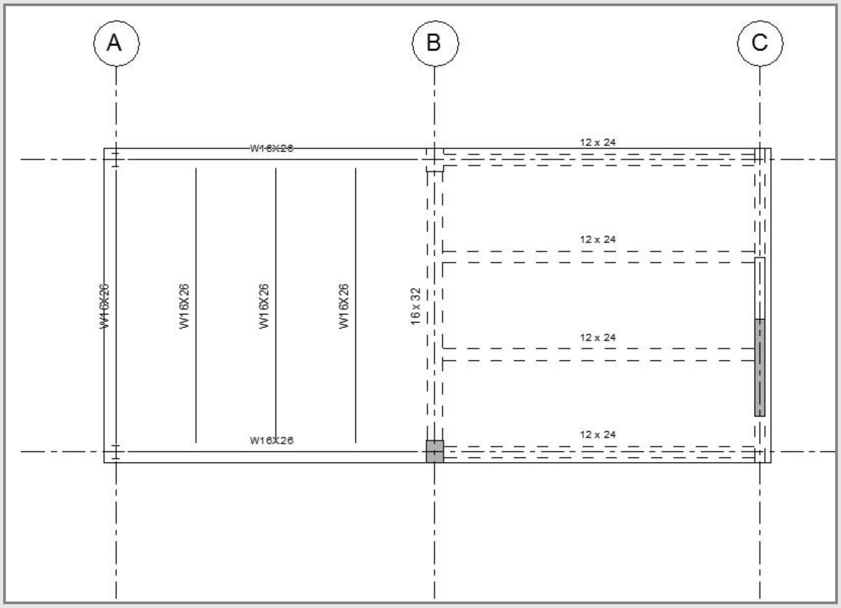
Referring back to Figure 17.15 for the Structural Columns category, if shown in a view (not cut), the primary control comes into play. Structural Columns – Projection will define the pen weight. The color on that row will set the visible color, just as the line pattern defines the line type to be used. In the case of columns, since there are many types of columns you should not set material here as it will be used everywhere unless an instance material has been set.

The other Structural Columns category controls are fairly simple as well. Hidden faces come into play when a face of a column is hidden by itself. An example is a wide flange column in an elevation turned with the flange facing the view. The line work for the hidden web is a hidden face. A concrete column that stops below a slab in a plan view is a hidden line. The Structural Column – Stick Symbol control applies to steel columns in coarse view detail level.

Each object category has its own set of controls, but many are similar and follow predicable rules. It is this consistency that makes object styles the best way to control your graphic standards for a project model. Earlier in this chapter you learned about line weight and patterns. This is where you should apply those standards first. Whatever changes you make here apply to the entire model and all views that don't have overrides.

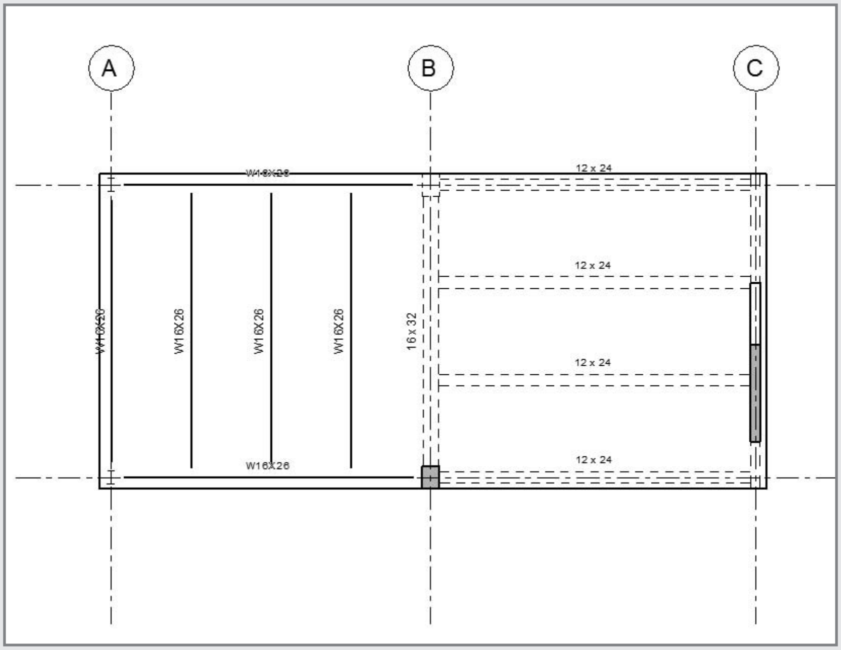
Exercise: See the Difference

This exercise will show you the difference between an out-of-the-box model appearance and an adjusted object styles model and how quickly it can be accomplished. Open `Chapter 17 Model Project.rvt` (from the book's companion web page). This model has a basic structure laid out showing a number of objects types. I have already created my preferred line patterns and line weights and I've set the concrete objects to have a solid shade hatching.



1. Choose Settings ➢ Object Styles.
2. Change the Floors Primary Projection Pen Weight from pen 2 to 7.
3. Scroll down to Structural Columns and expand the category.
4. Change the Line Weight Cut for the primary control from pen 4 to 8.
5. Change the Hidden Lines Projection Line Weight from pen 1 to 6. Also change the Line Pattern from Dash to #Hidden 1⁄16″.
6. Scroll down more and expand the Structural Framing category.
7. Change the Structural Framing Girder Projection control from pen 6 to pen 10.
8. Change the Hidden Lines Projection control from pen 1 to 5 and change the Line Pattern from Dash to #Hidden 3⁄32″.
9. Change the Structural Framing Joist Projection control from pen 4 to pen 10.
10. Scroll down to Walls and expand the category.
11. Change the Walls Primary Projection from pen 2 to 7 and Cut from 5 to 8. Change the Walls Hidden Lines Projection from pen 2 to 6. Change the Line Pattern from Dash to #Hidden 1⁄8″.
12. Click OK to apply the changes and exit the dialog box.

Now as you can see in the following illustration, all visible model objects changed graphically (hopefully for the better).



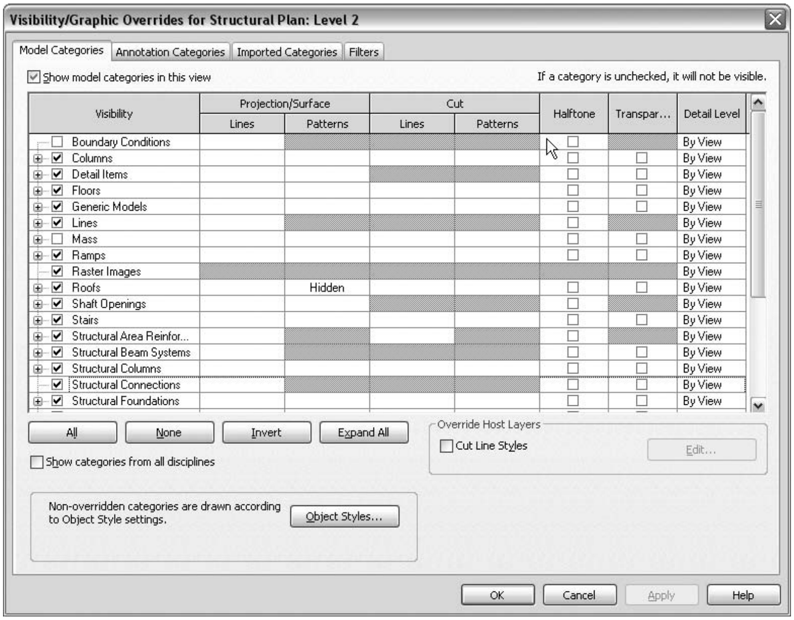
The key to making these adjustments is knowing your standards so that you can apply them. Print out a list of your pen weights along with sample sheet of line work so you can see what you are about to assign.

Visibility/Graphics

There will come a time when a given view needs a different set of object styles. It could be because the view scale is too large. In such cases you should utilize the differing view scale pen weights as discussed earlier in this chapter. But what about for specific changes, such as making floors have a surface pattern, or hiding beams, or making some items halftone?

Enter Revit Structure's biggest dialog box, Visibility/Graphic Overrides. As you can see in Figure 17.16, this dialog box looks somewhat intimidating. Not only does it provide controls for the same object options shown in the Object Styles dialog box, but you can even set some object categories to halftone, transparent, and a different view detail level. Additionally, using filters you can change only specific objects to look different, based on object type or just those selected objects.

FIGURE 17.16
The Visibility/Graphic Overrides dialog box allows you to customize the appearance for specific views.

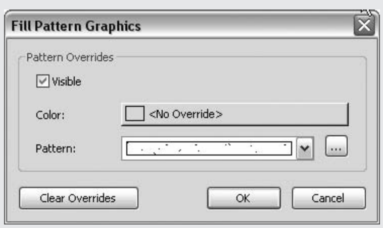


When you go to use the Visibility/Graphic Overrides dialog box, your purpose will probably be to solve some specific problem. Simply make your change and go about your other work. In some cases, the changes will require repeating for other views, in which case you can use the View ➢ Create View Template from View command. This command will save your changes into a view template, and you will be able to recall and apply this to other views, saving a lot of editing time.

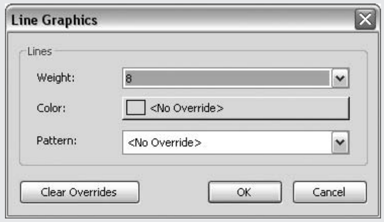
EXERCISE: CUSTOMIZE A VIEW

This exercise will show some view customization on the same model used for object styles. Open `Chapter 17 Visibility Graphics.rvt` (from the book's companion web page). You will see that there are some objects added to a filter in order to control them.

1. Choose View ➢ Visibility/Graphics to display the Visibility/Graphic Overrides dialog box.
2. Locate the Floors category and click in the slot for Projection/Surface Pattern. You will see an Override button; click it.
3. In the Fill Pattern Graphics dialog box, change Pattern to Sand and click OK.



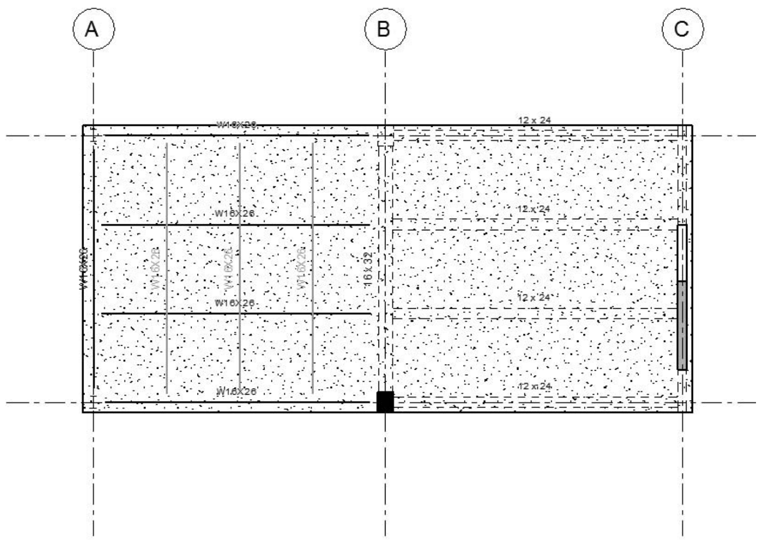
4. Locate the Structural Columns category and click in the slot for Cut Lines, and then click the Override button.
5. In the Line Graphics dialog box, change the Weight value to 8.



6. Click the Pattern slot for Structural Columns and change the pattern to Solid Fill.
7. Click the Filters tab at the top of the Visibility/Graphic Overrides dialog box. Click the Add button.
8. In the Add Filters dialog box, select Demo from the list and click OK. This then displays the Demo filter with a visibility control like the one for model objects.
9. Click the Halftone toggle at the end of the line. Click OK to apply and close the Visibility/Graphic Overrides dialog box.

As shown in Figure 17.17, a number of things changed. The floor got a fill pattern applied to entire surface. The columns now have a thinner pen weight and the concrete column got a solid black fill. Additionally, a group of steel beams set for demolition are now halftone

FIGURE 17.17
The finished changes applied to the model



Overall, you've made quite dramatic differences with fewer than 30 seconds' worth of edits to the Visibility/Graphic Overrides dialog box. You only have to use your imagination to know the boundaries of this capability. In general, making changes this way should be limited to specific views for specific needs. This will ensure that your project standards are model wide but you'll have a fallback solution when you need standards for specific views.

Color Modeling

Many users of Revit Structure products have had previous experience with AutoCAD. Therefore, typically the first emotion when a new user fires up Revit Structure and starts modeling is "I miss color." Revit Structure by default is black and white; all objects are white or black depending on your background screen color. There are few instances of color; levels that have views referencing them have blue target symbols. And sections and elevation views are blue as well. Reference plane lines are a green hidden line.

So why is there no color in Revit? Is black and white so much better than color and all those AutoCAD users have been doing it wrong for over 20 years? Revit Structure didn't start as an Autodesk-owned application. It was simply Revit and in many ways was competing with Autodesk products. The answer is, Revit Structure was made black and white so that it would *not* be like AutoCAD. Plain and simple, there was a marketing decision to be different and the only way to be different was to go no-color.

Now, when you made the transition from AutoCAD to Revit Structure, no doubt you just accepted the black-and-white display. But perhaps you have poked around a little and discovered that color is there—it's just not used. Color is tedious to apply, but once you experiment with it in a model, applying differing color to columns, walls, beams, and the like, it will begin to feel familiar. You will then notice something simple and yet profound: it is easier to tell what you are looking when it is in color!

Interestingly Revit Structure doesn't use color to control pen weight as AutoCAD can, so color inherently won't affect your look unless you plot in color. But imagine you are zoomed into a tight area and you see a hidden line next to another hidden line. Which is a column and which is a beam? Well, you could hover your mouse above that line or select an object to know what it is, but wouldn't it be easier to simply know by the color? You would know visually whether a beam is aligned with the wall, or the wall is aligned with the beam—by the color of each and recognizing what is right and what is wrong.

The focus of this section isn't to propose a color scheme; that is up to you and your fellow modelers. What we suggest is to use color as an advantage in modeling with Revit Structure. It really can help.

What to Leave Black and White

Not everything in Revit Structure needs to be shown in color. For example, detail components you might draw in a drafting view have no 3D and gain no benefit. The same thing can be said for text in your model views; if you see text you obviously know what it is. A good rule to follow is that only model objects get color; all annotation and detail line work stay black and white.

There's a side benefit to making line styles only black and white: if someone uses the Line work tool to paint something, such as a solid line of a model edge to be a hidden line style, you will then see black and white on a model object, telling you that some model fudging was going on. You can then investigate and correct any issues so that the model will automatically display a hidden line rather than faking it.

The Bottom Line

Interpret what can and cannot be done easily. Standards are there no matter what. What they are and how they are controlled are up to you. Before Revit Structure, users had various files provided with AutoCAD that determine what most of the standards were based on. A standard is many things, but a basic one specifies line weight, pattern, and style. Additionally, you have filled patterns that can tone contained areas. You then bring all these definitions together and apply them to object styles, views, and objects to create your drawings.

> **Master It** Develop line weights that meet your needs. Create line patterns and styles and then apply them to your model via Object Styles. Address annotation and fill standards as well, all matching your required standards.

Enhance your model through customization. Anyone using Revit Structure deserves to have their tool be as productive as possible. With a little bit of practice, you can take using Revit Structure to a new level.

> **Master It** Anyone can add a new Split with a Gap command, as well as control the size of temporary dimension values. Follow that with gaining access to your firm's library files as well. Then improve usability by using command shortcuts and take advantage of input speed.

Implement model standards and view overrides. Once you have your standards in place, you then have to apply them to your model. There is no need to simply accept what Revit Structure can produce right out of the box.

> **Master It** Take your standards and use them to control your model. Then when the need arises, tweak just about anything for a single view at a time. Can you break out of the black and white box and think in color again?

Chapter 18

Family Creation: Beyond the Built-In Libraries

There are two ways you may have arrived at this chapter. One is you read through the book in its entirety and are now ready to start creating some families. The other way is you went out, bought the book, and flipped directly to this chapter with the sole purpose of learning this part of Revit Structure. Well, whatever path you used to get to this point, you now know that Revit Structure is very much dependent on family objects to drive the model. For all of the advantages of BIM and the awesome workflow that Revit Structure can provide, you will find that you are virtually going nowhere unless you have the content you need. Yes, Revit Structure does provide a rich library of content that will get you through a project, but what about the specialty content you need? It would literally be impossible for Revit Structure (or any CAD application for that matter) to predefine a custom family for every situation and to be able to re-create that special family exactly in the way your company's standards are set up.

Revit Structure acknowledges this, so the approach is different. Instead of trying to provide thousands of blocks, cells, and little programming routines, Revit Structure provides blank templates (`.rvt`) that allow you to choose the basic type of family object you want to make. Each family in Revit Structure is built with the flexibility to adhere to as many situations as possible. For example, say you have to add a lintel into your project. In AutoCAD, you have a block that is inserted into your drawing. You then explode, stretch, and modify that block to suit your specific dimensions. Then you add another block in plan to call out the lintel (usually with a single hidden line and a text mark). In Revit Structure, you make one lintel family starting with a Revit Structure family template, with parameters that allow you to change the size, shape, and anything else you build into the family. You can also create the family to display the line work in plan. Taking it a step further, you can even get the family to show up in schedules and material takeoffs.

Revit Structure is truly driven by families. Some of the families you create will be quite simple once you get the basic rules under your belt. Others are going to be more difficult and will require more of a thought process and good old-fashioned patience. Either way, learning Revit Structure families is not accomplished overnight. Rome was not built in a day, but little parts of it sure were.

In this chapter you will learn to:

- Create a footing step family
- Create in-place families
- Create groups

Creating Families

In Revit Structure, a family is named *family* for a few reasons. One is the hierarchical concept that drives the functionality, and the second is in the delivery of the family to a project. It is similar to when a minivan pulls up to the picnic area of a local state park and 12 people, all parents and siblings, start piling out of the vehicle. One van, 12 people. The same concept holds true for a Revit Structure family.

In Revit Structure, there are three different types of families:

System family This is the type of family you have been using all along. A system family is defined within the model and is not exported or inserted. When you edit a system family, you either edit the information provided in the Element Properties dialog box, or Revit Structure brings you into a sketch mode. The items contained within the model that are system families are:

Walls

Wall footings (strip footings)

Floors (slabs)

Roofs

Stairs/ramps/railings

Toposurfaces

Openings

In-place family This type of family resides within the model as well, but when you issue the Create In-Place Family command, the Design bar switches to the Family Editor. You can then create the family components in the position they need to be in without having to insert them or position them correctly. Also, in many cases you need the other model geometry to actually lay out the family. It is imperative that this functionality be allowed in Revit Structure.

Family files (`.rfa`) This type of family is created and managed externally and then inserted into the Revit Structure model. For instance, if you wanted to add a castellated beam into the model, you would have to browse for one. By default Revit Structure does not have one of these families preloaded into the model. You start the Beam command and then click Load on the Options bar. This allows you to browse for the Revit Structure family. In the interest of keeping the size of your models to a workable level, it is nice to have a library of these families available but not loaded in the model until you actually need them. This topic will be covered in-depth later in the chapter.

When you construct a family in Revit Structure, it is a unique process very much unlike any CAD application. At first you may find the family-creation process to be cumbersome and somewhat unwieldy. After a while and some practice, you are going to see the overwhelming advantage you have by constructing these components. To get a feel for the creation of families, we will use a stepped footing example. It will be similar to the stepped footing you have already used in Chapter 8.

Creating a Family File

To begin creating a family file (`.rfa`), you should first find a template that is close to what you are trying to create. Revit Structure provides templates for most situations. For example, if you were looking to design a truss, you would start with the `Structural Trusses.rft` file found in the Imperial Templates directory. Once you have this file open, you will see that the Design bar now is the Truss Family bar, as illustrated in Figure 18.1, and it provides functionality specific to the design of a truss. Also, specific materials are preloaded into these files.

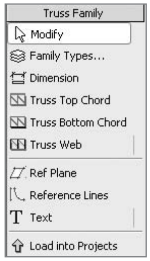


FIGURE 18.1 An example of using a specific template for the Structural Truss template

The process for starting a new Revit Structure family is to choose File ➤ New ➤ Family from the Revit Structure menu. This will launch the New Family - Select Template dialog box. You can now browse the Revit Structure Imperial Template library for your specific family. For the examples in the book, choose the `Structural Foundation.rft` template file (`.rft`), as shown in Figure 18.2.

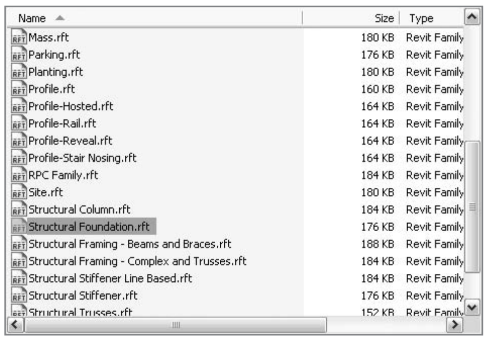


FIGURE 18.2 When you choose File ➤ New ➤ Family, the New Family dialog box appears, allowing you to choose from one of the predefined Revit Structure families.

Once you are in the desired template, you are basically on your own. As you can see, you are given very little graphical information to begin with. If you type WT (or choose Window ➤ Tile), you will see that Revit Structure opens four views simultaneously. These are a plan, a front and left elevation, and a 3D view. You will most certainly use all of these views at some point, but for now, to get started only use the Floor Plan: Ref. Level view. Notice that there is a Project Browser as well. This allows you to navigate through the family just as you would navigate through a Revit Structure project.

With the floor plan open, notice the two green dashed lines. These are reference planes. Depending on the template used, Revit Structure often will start you with at least two of these in

plan view and usually a datum reference in elevation view. When you build a family, these reference planes are the key to success. They are used heavily in the effort to constrain the family, allowing only user input to determine the resulting geometry. To make a long story short, reference planes help define the family and control placement in the host model.

Reference Planes in Families

In Figure 18.3, you will see two reference planes. These two reference planes are important, and you should always build from them. You should not modify or move these planes, as they define the origin and the placement of the family. For example, select one of the reference planes. You will see a label, as shown in Figure 18.3. This indicates that the reference plane has a specific purpose. With the reference plane selected, click the Element Properties button on the Options bar. You will see a check box next to Defines Origin. This further indicates that these two reference planes have a special value and should be used as a building block.

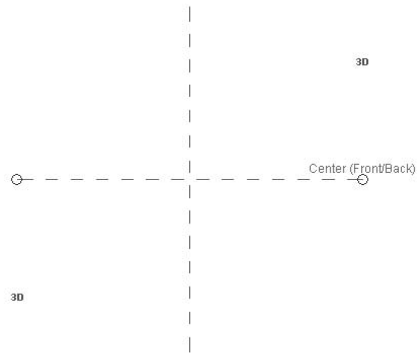


Figure 18.3
These reference planes define the origin of the family and should only be used as a starting point to build from.

The family we are about to create is the footing step illustrated in Figure 18.4. The objective is to make the family dynamic enough to fit into any stepped situation. This means that the dimensions and geometry need to be flexible enough to allow the component to be placed manually and then adjusted to fit.

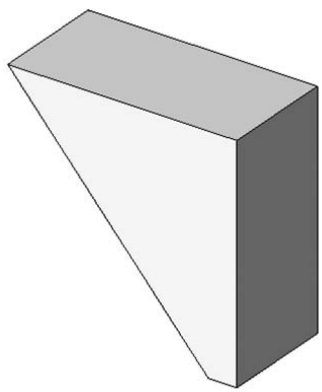


Figure 18.4
The final footing step family

To create these dimensional constraints, you will need to add additional reference planes offset from the existing reference planes you see in Figure 18.3. To do so, focus your attention on the Design bar, which is now reduced to one tab labeled simply Family, as shown in Figure 18.5. On the Family tab of the Design bar, you will see a Ref Plane button. Click it.

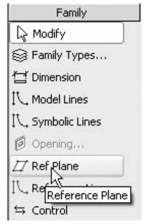


FIGURE 18.5
The traditional Design bar contains only one tab in Family Editor mode. This is the Family tab. Most of your commands can be launched from here.

Once the Ref Plane command is running, you will notice that the Options bar becomes populated with choices for the Ref Plane function. By selecting the Pick Lines button and adding an offset of 4′-0″, you can now click the vertical reference plane in the view window. When you hover the mouse over the reference plane, you will see a blue alignment line indicating the direction in which the reference plane will be offset. Once the blue alignment lines appear to the left of the vertical reference plane, select the vertical reference plane. Your view should now look like Figure 18.6.

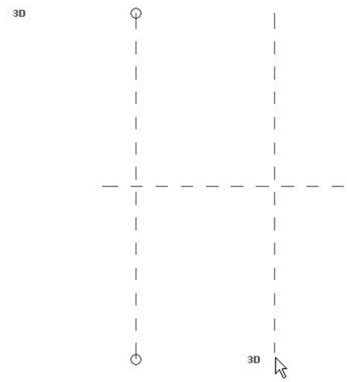


FIGURE 18.6
Adding a new reference plane allows you to construct the family "stick by stick."

The concept here is that when the footing step is inserted into the model, the insertion point will be the right midpoint of the family in the vertical plane. The horizontal plane, however, should be centered on the footing. To create this scenario, you need to add two more reference planes above and below the horizontal reference plane. To do this, start the Ref Plane command,

and offset the horizontal reference plane up 1′-0″ and down 1′-0″, creating three horizontal reference planes, as shown in Figure 18.7.

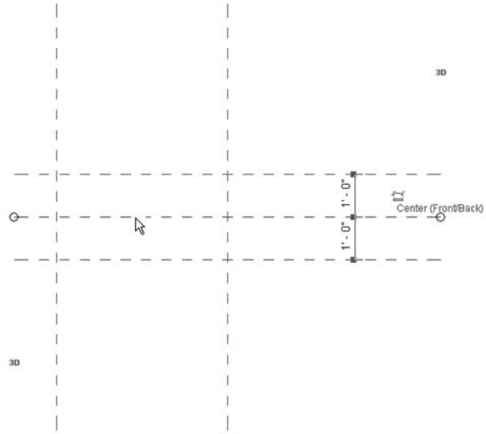


FIGURE 18.7
By adding two additional reference planes, you have centered the footing on the horizontal reference plane.

At the bottom of the footing it will be nice to have an extension, which will provide more bearing surface against the bulkhead of the wall step. Normally this is a 6″ extension of the wall footing. We can build this right into the footing step. Start the Ref Plane command, and offset a reference plane to the left 6″. This will establish a plane for any additional cover you may wish to add to the step. Later on, you will create an extrusion that will be locked to these planes. When the reference planes move, the shape will be altered.

In plan view, the most of the footing step has been defined. Now it is time to move to another view. In the Project Browser, notice that there are four elevations predefined. Choose the front elevation. You can now see a datum reference and two of the reference planes established in the plan view. You now need to add a height to the footing step. To do this, follow the same procedure as you did in plan view. Start the Ref Plane command, and offset the datum level up 4′-0″. You may notice that there is an elevation line as well as a reference plan at 0′-0″. It is good practice to use the datum line to create the new reference plane to offset the new plane. The reason for this is that the datum line is the strongest reference in the component. This will prevent the family from behaving erratically when placed in the model. See Figure 18.8.

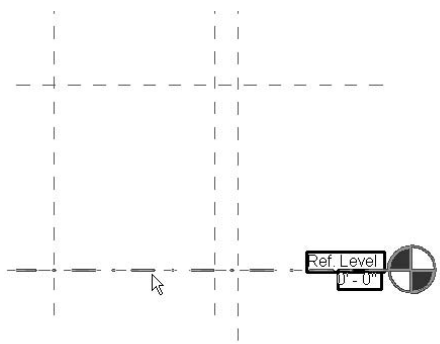


FIGURE 18.8
Establishing a height reference plane is done in elevation view exactly the same as it is done in plan view. Remember to use the datum line as the basis of the offset.

Since this is a pretty simple family, these are all the reference planes we need. It is good to look at this procedure exactly the same way as you would look at the procedure for creating a steel-framed structure for the building. These reference planes are the structure of the family. Any additional items we add are always going to be locked to these planes. In the model, you will not see the reference planes we just added, but the massing items such as the concrete step will be supported and influenced by the control of the planes.

Now that the reference planes are in place, it is time to add some parameters and labels to them. When you do this, the end user can go to the element properties of the footing step and type in new dimensions, creating different configurations of the same footing.

Parameters and Labels

Yes, if you are thinking this is the fun part, then you are correct! Adding flexibility to the family is precisely the functionality Revit Structure provides that is going to make you never want to go back to a CAD-based platform again. Fundamentally this procedure is simple. Of course, when you get into more complicated families, it can get harder, but the base concept is grounded in simplicity. You are going to dimension these reference planes and then add labels to those dimensions. Those labels become parameters that the user sees in the Element Properties dialog box. For instance, you added a 4′-0″ reference plane in the front elevation. The next step is to add a 4′-0″ dimension and label it Height. Now, the user can change the Height parameter to 5′-0″, and viola! It is now 5′-0″ high.

To add dimensions, go back to the Ref. Level plan view. Again, one really nice thing about the Family Editor is that the Design bar is reduced to a single Family tab. On the Family tab of the Design bar, you will see a Dimension button. Click it, and add a 4′-0″ dimension, as shown in Figure 18.9. Be sure to *start* the dimension from the center (left/right) reference plane. Remember the two reference planes that we used to start from. Always dimension from these planes. You will find you will have a much more stable family when it comes time to place the new component into your model.

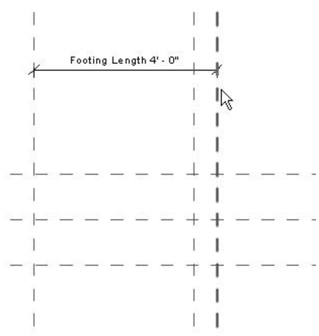


Figure 18.9
Dimensioning the reference planes is the link between the physical geometry and the element properties.

After the reference plane is dimensioned, you can then select the dimension. As is true for any Revit Structure component, once the item is selected, you will see choices on the Options bar specific to the selection. In this case, there is a Label menu. You can select the <Add Parameter> item from the drop-down list, as shown in Figure 18.10.

Figure 18.10
Once the dimension is selected, you can add a parameter from the Options bar.

The next dialog, as illustrated in Figure 18.11, has a lot of power behind it in spite of its simplistic appearance. Some of the choices you make here are irreversible, so be deliberate when you are creating these parameters. In the Parameter Properties dialog box, the first set of choices deals with the type of parameter you wish to use: Family Parameter and Shared Parameter. Each has a specific purpose.

Family Parameters

If you choose Family Parameter (which is the default), that means this parameter will occur only in this specific family and not in any other. Given the specific nature of this type of family, it would be impossible to allow it to show up in schedules and tags because it is specific to only one family. You would wind up with a separate schedule and a separate tag for each instance of the family—not good. The reason to choose this is to avoid having literally hundreds of useless parameters embedded in every family of this type. The functionality of Revit Structure would also take a huge performance hit if this were the case. Having family parameters allows you to make any parameter you see fit within this family without affecting any other item.

Shared Parameters

If you choose Shared Parameter, Revit Structure will look for a `.txt` file, where parameters are stored. This text file is read by both the model and the family. You will be adding shared parameters later in this chapter. For now, choose Family Parameter, as shown in Figure 18.11.

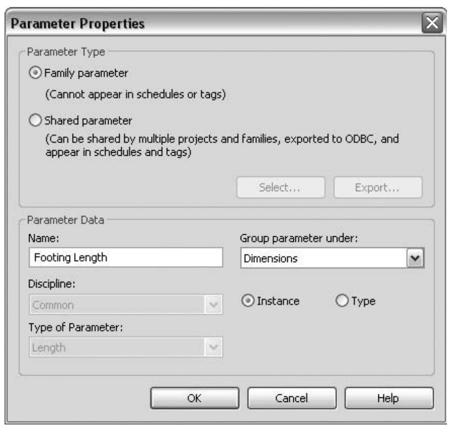


Figure 18.11
The choices made in the Parameter Properties dialog have a large influence on the behavior of the family.

Parameter Data

The first item in the Parameter Data section of the Parameter Properties dialog is Name. Remember that you accessed this dialog after selecting the 4′-0″ dimension. The parameter name should make sense as it relates to the item you are labeling. Name this **Footing Length**. Notice the name Footing Length contains capital letters. This comes into play if the parameter is later added to a formula, and then it becomes case sensitive. Be careful with this one as it could become quite bothersome. If you choose to capitalize your parameters, be consistent.

Notice that the following two data fields are filled out and are unavailable for modification: Discipline and Type of Parameter. They are uneditable based on the fact that Revit Structure knows you chose a dimension, so the Discipline is automatically going to be Common, and the Type of Parameter is going to be Length. Again, both of these values can make a huge difference if the parameters are involved in a formula.

The Group Parameters Under drop-down list allows you to categorize the parameters under a specific heading in the Element Properties dialog box. This type of length parameter is almost always grouped under Dimensions.

The next two choices are Instance and Type. This is another big choice. If you choose for this to be an instance parameter, you are saying, “I want this parameter to apply only to the component I have selected.” For example, if you select the footing step after you insert it into the model and then open the Element Properties dialog box, you will see a list of parameters. If you change any of these parameters without clicking the Edit/New button, only the footing step you have selected will be modified based on the changes you make. The rest of the footing steps will remain unmodified. If you choose a type parameter, you must first select the footing step in the model. Then click the Element Properties button, and select Edit/New. Now if you make changes to these parameters, you will change all of the footing steps in the model. For the footing step, make all of the parameters instance parameters.

Now if you click OK, the dimension will contain the new label, as shown in Figure 18.12.

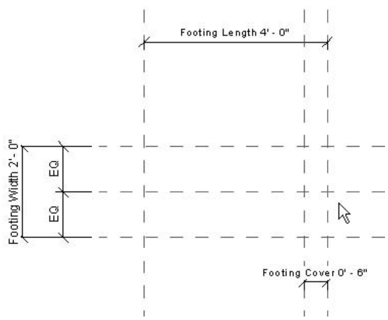


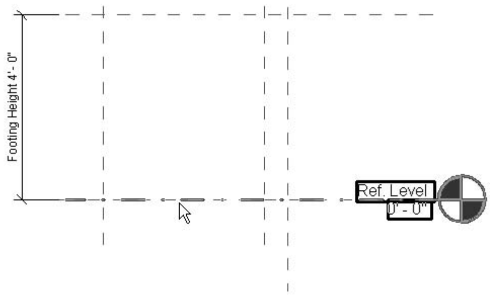
FIGURE 18.12
The reference planes now have the added dimensions and labels. Notice that the first two center reference planes are still being used as the origin.

Now it is time to deal with the two 1′-0″ spaced reference planes you added earlier. These reference planes collectively will represent the width. You will, however, need to tell Revit Structure that although the width needs to vary, it always needs to be equally spaced about that center reference plane. To do this, start the Dimension command, and place a two-dimension string including all three horizontal reference planes. Once you do this, click the blue EQ icon. This will lock the reference planes relative to the center reference. Now you can add one more dimension to the overall 2′-0″ width, select it, and add a parameter called **Footing Width**. Make sure

the parameter is grouped under Dimensions and that it is an instance parameter, as shown in Figure 18.12. While still in plan view, dimension the 6″ reference plane and give it a parameter called **Footing Cover**.

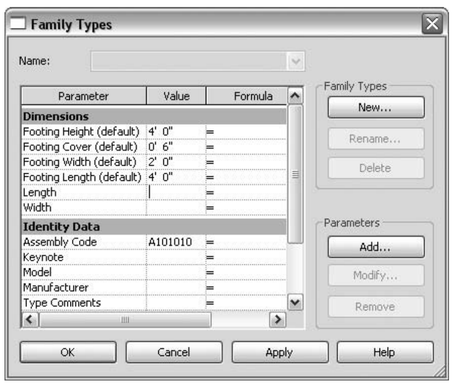
Are you starting to see the process here? You build the geometry with reference planes, and then you dimension and add parameters to them to make them flexible. To continue, go to the front elevation where you added the 4′-0″ reference plane based on the datum elevation. Add a dimension starting at the datum line to the top reference plane, as illustrated in Figure 18.13. At this point the procedure is the same as in plan view. You can select the dimension and add a new label to it. This time name it **Footing Height**.

FIGURE 18.13
The Footing Height dimension and parameter are added.



It is now time to start testing the family before you get too far. The reason for this is that if your first test is when you actually insert the family into the model and it breaks, it could be very difficult to figure out exactly where the family failed. Once you have established the majority of the reference planes and have added parameters to them, you can start to flex the family to shake out unexpected results. To do this, click the Family Types button on the Design bar to bring up the Family Types dialog box shown in Figure 18.14. Once you have the Family Types dialog box open, you can see the variables that were added as you were labeling the dimensions. Notice that the category for all of the parameters is Dimensions. You may also notice that each parameter name has “(default)” next to it. This indicates that it is an instance parameter, and the value applied here is simply the default value as it is inserted to the model.

FIGURE 18.14
To access the Family Types dialog box, click the Family Types button on the Family tab of the Design bar.



These parameters can be edited. If, for instance, you forgot to make a parameter an instance parameter, you can fix it by selecting the parameter in question and clicking the Modify button in the Parameters field to the right of the dialog box, as shown in Figure 18.15. This puts you back in the Parameters Properties dialog box, where you can make the changes accordingly. For this footing make sure that all of the parameters you added are instance parameters. If some are not, you will be unable to add them to an equation or a formula.

FIGURE 18.15
First select the parameter you wish to modify, and then choose one of the action items from this list.

To test the model, you will make a modification to the value established to the right of the parameter name. In this case, you can change the Footing Width value to **1′-6″** and the Footing Length value to **5′-0″**. Click Apply and then OK. Your reference planes will move away from the origin reference planes. This is called "flexing the family." Do this procedure as often as possible.

Now, suppose you wanted the Footing Cover variable to be a percentage of the Footing Width value. You can do this as a formula in the Family Types dialog box. To do so, open the Family Types dialog box, and find the Footing Cover variable. To the right of the Value column, you will see a Formula column, as shown in Figure 18.16. Within this column, Revit Structure will accept a formula. For the formula, type **Footing Width / 3**. Now if you change the Footing Width value, the Footing Cover value changes.

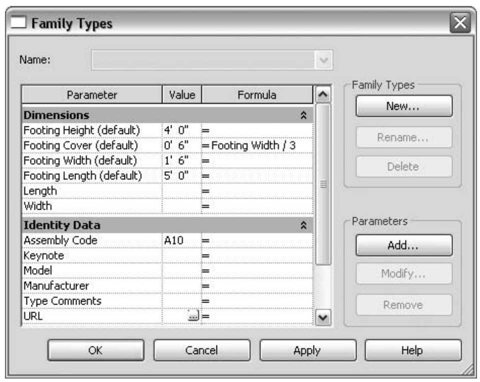


FIGURE 18.16
You can add a formula to the Formula column in the Family Types dialog box. Remember that the expressions are case sensitive, and the parameters must be of the same type (instance or type).

Now that the structure of the family is firmly in place, it is time to add the physical mass that you will see in the model. The extrusion that you will add is simply locked to the reference planes that you have established.

Adding Extrusion Geometry

This is the fun 3D part. When the family is flexing correctly, you can usually wrap this up with adding a simple extrusion or two. The hard part is pretty much over at this point. There can, however, be a few tricky methods to deal with. The first is establishing the plane to which the extrusion is affixed. This extrusion will be created in the front elevation. When you start the Solid Form ➢ Solid Extrusion command, Revit asks you at which plane you wish to start the extrusion. You want the back plane, but it does not show up as a choice in the dialog box. At this point, it is a good idea to start labeling the reference view and click the Element Properties button. In the Element Properties dialog box you will see a Name field. Name the reference plane **Front**. You can then repeat the Element Properties dialog box, and name it **Back**, as shown in Figure 18.17.

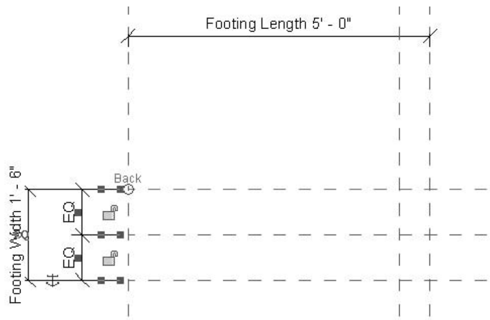


FIGURE 18.17
This plane has been given a name. To name a plane, select the reference plane, click the Element Properties button on the Options bar, and add a name to the Name field.

To add an extrusion, it is appropriate to go to the front elevation. One good thing to remember while adding an extrusion in Revit Structure is that the extrusion is always coming toward you. In this case, you would want to start the extrusion on the back plane so the resulting geometry fills the area between the two reference planes.

Switch to the front elevation. On the Family tab of the Design bar, you will see a Solid Form button. When you click this button, a small menu will appear giving you the choice of extrusion to use. Choose Solid Extrusion. Immediately Revit Structure will ask you which plane you wish to use. In the drop-down list, select Reference Plane : Back, as shown in Figure 18.18.

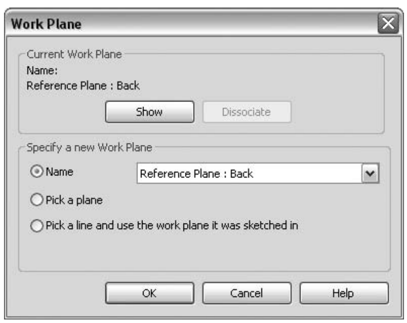


FIGURE 18.18
It is important to have a clear naming strategy for defining extrusion base points. The new reference planes become available if they are named.

Of course the Options bar plays an important role here. On the Options bar click the Pick Lines button, and make sure Lock is toggled on. It is best to select the reference planes where possible. Once you have selected the reference planes, you can click the Draw Lines button and sketch in the diagonal line. Be sure you are snapping this line to the reference plane intersections. See Figure 18.19. Once you have the perimeter of the extrusion sketched in, use the Trim command to clean up the corners.

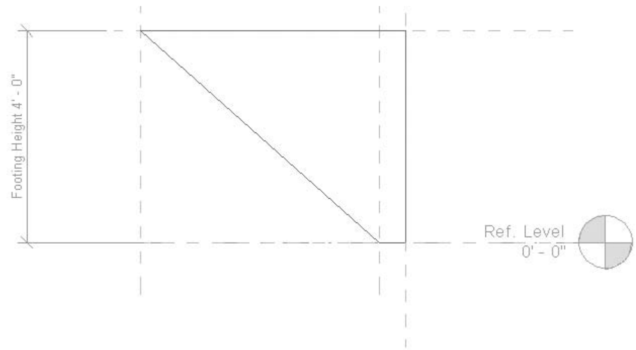


FIGURE 18.19
To draw the perimeter of the extrusion, you should select the reference planes and lock the sketch lines. When you are sketching lines without the benefit of having a reference plane, be sure you are snapping the ends to the reference plane intersections.

Now that the perimeter is sketched and locked to the reference planes, you need to specify a material and an extrusion depth. On the Sketch tab of the Design bar, you will see the Extrusion Properties button. Click it, and in the Extrusion Properties dialog box you will see two fields that deal with the extrusion's constraints: Extrusion Start and Extrusion End, as shown in Figure 18.20. Since you know that the Extrusion Start refers to the back reference plane, you know you can keep this value at 0. The end, however, needs to be the same as the Footing Width parameter. If you notice, you could simply type the increment in the Value field, and the extrusion would look okay. But when you flex the model, the extrusion depth will remain at that fixed value you specified. To add the Footing Width parameter to this value, look to the right of the Value column. There is an = column. Within the = column you will see a small gray button, as pointed out in Figure 18.20. Click this button, and you will get a list of length parameters to choose from. Select Footing Width, and now the extrusion will be whatever length this specific parameter contains. Now the small gray button shows an equal sign, and the field is grayed out.

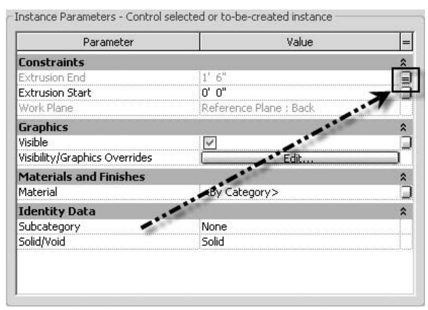


FIGURE 18.20
Adding parameters to an extrusion is done through the small button to the far right of the row.

While still in the Extrusion Properties dialog box, navigate down to the Material field. The value for the material will be <By Category>. Since these types of masses are almost always cast-in-place concrete, it is safe to specify this as a material. If you click the <By Category> field, you will see a small […] button appear. Once you click this button, the Materials dialog box will open. Here you can select Concrete - Cast-in-Place Concrete. Once you return to the Extrusion Properties dialog box, you will see this value added, as shown in Figure 18.21. If you do not assign a material, one will be automatically assigned using the material for the category that you specified for your family.

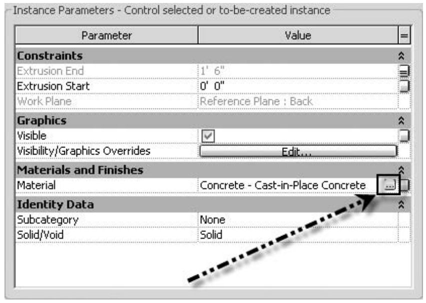


FIGURE 18.21
You can add a material by clicking the small Builder button in the <By Category> field.

Once the Extrusion End and the Material fields are satisfied, it is safe to close this dialog box. On the Sketch tab of the Design bar, click Finish Sketch. Go to a 3D view to be sure the shape is the way you expected it to be. At this point it is strongly recommended that you flex as many parameters as possible while viewing the family in 3D, as shown in Figure 18.22.

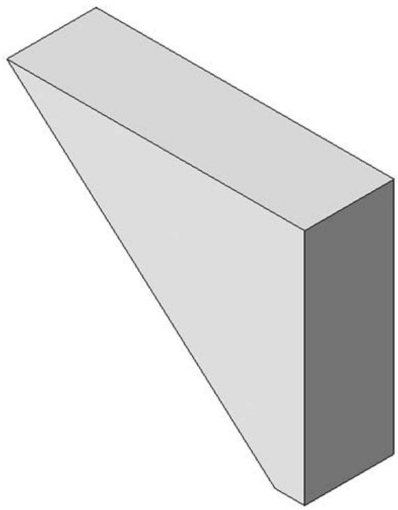


FIGURE 18.22
The finished family should be flexed as much as possible to test for unexpected behavior.

NAMING THE FAMILY AND SPECIFYING FAMILY TYPES

You will physically name the family by saving the file to your network with a specific name. If you have not done so, click the Save button, and put the footing step file onto your network. The

footing step file is named `Footing Step.rfa`. You can download it from the book's website at `www.sybex.com/go/masteringrevitstructure2009`.

You can name yours the same. Now when you insert this footing step into your project, it will show up in the Type Selector as the family's filename, as shown in Figure 18.23.

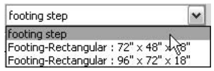


FIGURE 18.23
In the actual project, the footing step family is indicated as such by using the filename in the Type Selector.

One thing more to do—check the Family category. It would be nice to be able to insert this family from the project's Design bar as an isolated foundation as opposed to a general component. The ability to do this not only makes the process more organized but gives you a better chance of this family automatically joining to adjacent foundations. To add or check on the family's category, choose Settings ➢ Family Category And Parameters. Notice in Figure 18.24 that this specific family is already categorized as Structural Foundation. This is because of the template you used when you created the family. If you had simply used a generic model, you would have to manually qualify this family as Structural Foundations.

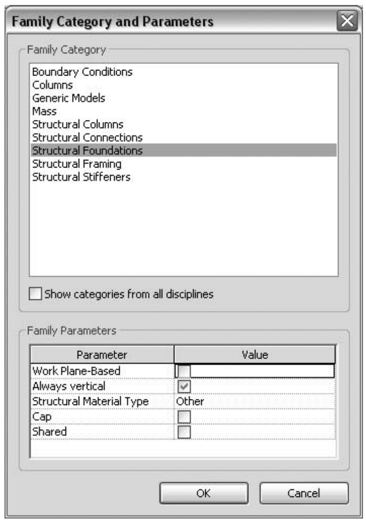


FIGURE 18.24
To make sure you can insert this family as a foundation, choose Settings ➢ Family Category And Parameters.

To add the family to a model, you need to either start a new project or open an existing one. With the project file open along with the family file, switch to the family file. The project will be open, but in the background. Within the family file (`Footing Step.rfa`) click the Load Into Projects button at the bottom of the Family tab. The project will now display. Now you can go to the Modelling tab on the Design bar and select Foundation ➢ Isolated. You will see Footing Step as

one of the choices in the Type Selector on the Options bar. Notice that this family is one of three. Also note that your footing step family looks a little different from Footing-Rectangular: 72″ x 48″ x 18″. This is because the Footing Rectangular family has multiple types assigned to it. To assign a type to the family, open the family file again, and click the Family Types button on the Family tab. Change Footing Height to **6′-0″**, Footing Depth to **3′-0″**, and Footing Width to **6′-0″**. In the upper right-hand corner of the Family Types dialog box, you will see the Family Types category, as shown in Figure 18.25. Click the New button. Label the new type **6′-0″ x 6′-0″ x 3′-0″**. You now have a new family type. Click New again, and add **5′-0″ x 5′-0″ x 2′-0″**. Change the parameter values to reflect these new sizes.

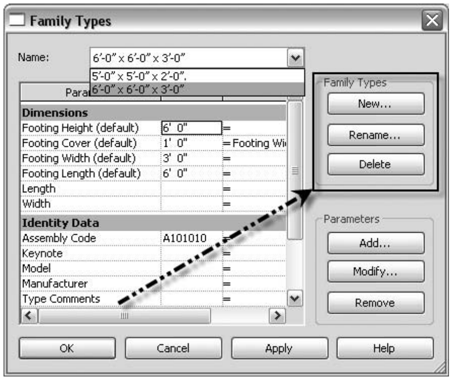


FIGURE 18.25
By adding new types, you can switch between different size configurations.

Once you have finished, you can load this file back into the project. Within the project, you can insert the foundation step the same way as you did before. You now have two variations of the same family: Footing Step: 6′-0″ x 6′-0″ x 3′-0″ and Footing Step: 5′-0″ x 5′-0″ x 2′-0″.

Real World Scenario

WHEN TO STOP!

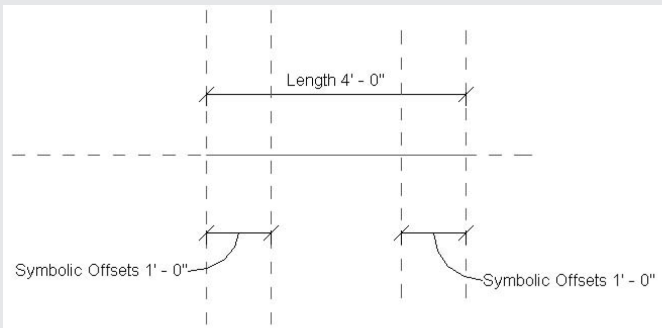
It is sometimes difficult to know where to draw the line as far as trying to capture every size configuration in a family type. Typically you can create a family and provide the end user with two or three size configurations to start off. Once the user adds the family to her model, she can then select the custom family, choose Element Properties, click Edit/New, and then select Duplicate to create a new family type. In the case of the footing step family, once the user selects the step, she will see blue shape handles appear. Without even going to the element properties, she can simply stretch the footing step to match and automatically join to adjacent foundations.

Some families you make are going to very simple. Some families you will attempt to make are going to take “Ninja” skills. Either way, it is quite obvious that the more you learn about creating good, intelligent families, the more benefit you will reap by choosing to use Revit Structure. In many cases, success in Revit Structure and BIM in general depends on how well you do creating families.

Exercise: Creating a Bridging Family

The following activity involves creating an uplift/joist bridging family. It will include family creation concepts along with practical uses and applications. This family will function by being drawn in a roof framing plan as a single line. The graphical result will be symbolic line work representation in plan view. In section view, there will be an angle seated on the bottom of the joist for detailing purposes.

1. To get started, open Revit Structure and choose File ➢ New ➢ Family. In the New Family dialog box, select `Generic Model line based.rft`. Notice you are given three reference planes, a predrawn reference line, and a labeled dimension of Length. You will create a profile extrusion along this reference line. The Length parameter will be a variable that the user specifies graphically as he draws the bridging in plan view.
2. In plan view, select the Ref Plane command, and create new reference planes to the left and the right of the existing vertical reference planes; each should be in 1´-0˝ toward the center of the 4´-0˝ dimension.
3. Dimension the two reference planes.
4. Select both new dimensions and add a label to each. Call the new label parameter **Symbolic Offsets**, group it under Graphics, and make it an instance parameter.

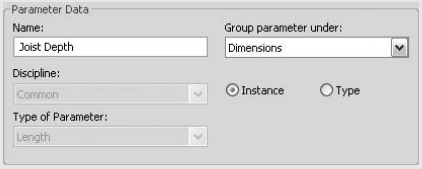


5. On the Family tab, click the Family Types button.
6. Change the Length value to **8'-0"** and click Apply. You can close the dialog box. If the right side extends 4´-0” and the Symbolic Offset reference plane went with it, you are good to go.

Select the reference plane to the far left. Notice that it is already labeled. This is a good thing because some of the logistical work has been done for you. This is the base origin of the entire family, so therefore this is where we want to start our extrusion.

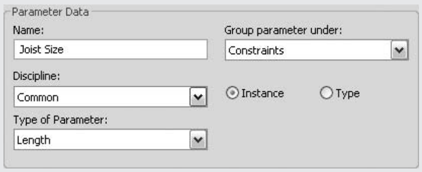
1. In the Project Browser, go to the Left elevation.
2. Start the Ref Plane command, and create a new reference plan down 1´-6”. This will be the elevation that the angle will sit upon.
3. Dimension the new reference plane. Be sure to start the dimension from the datum line.

4. Add a label to the dimension by selecting it and using the Label menu from the Options toolbar. Name it **Joist Depth**, choose Dimensions in the Group Parameter Under drop-down list, and make it an instance parameter.

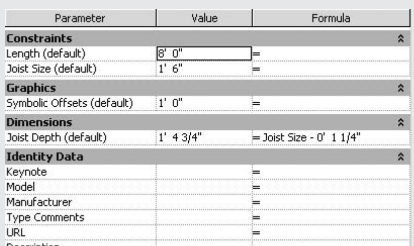


5. Click the Family Types button.
6. To the bottom right of the dialog is a Parameters field. Click the Add button.

7. Name the parameter **Joist Size**, under Type of Parameter choose Length, group it under Constraints, and make it an instance parameter.

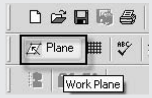


8. Click OK.
9. In the Family Types dialog box, give the new Joist Size parameter a value of 1´-6˝.
10. For the Joist Depth formula, type in Joist Size - 0’ 1 1/4” . The Joist Depth value will now change based on the added formula.



11. Click Apply and then OK.
12. Select the bottom reference plane (the one 1´ 4¾” below the datum).
13. Select Element Properties from the Options bar.
14. Name the plane **Bottom Reference**.

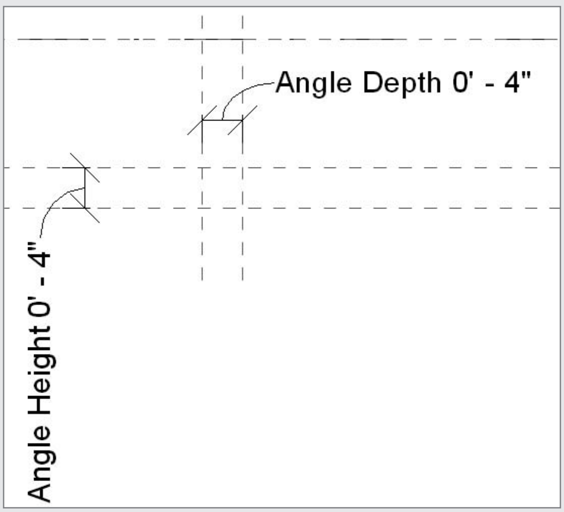
15. Choose the Ref. Level floor plan.
16. Toward the top of the Revit Structure dialog box, just below the save icon, you will see a Plane button. Click it, and set the work plane to Bottom Reference.



17. On the Family tab, click the Reference Lines (not Ref Plane) button. Draw a line from the left reference plane to the right reference plane.
18. Choose a 3D view to make sure you see two reference lines.
19. Flex the family by changing the Length value to **9'-0"**. If both lines stretch, you are good to go.
20. Choose the Left elevation.

At this point you can now draw some reference planes for the actual angle. This will only involve adding two additional reference planes, but it is here that you should start thinking, "Should I add more reference planes to provide additional functionality such as angle thickness?" There is no need to do so at this point, but when you are designing a family, always keep this in mind. It will take longer as the families become more complex, but the time you spend adding reference planes will certainly pay off later.

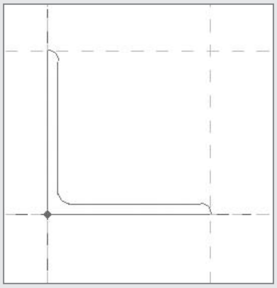
1. Start the Ref Plane command, and offset one reference plane up 4" and one to the right 4".
2. Dimension the two new reference planes.
3. Add a label and name it **Angle Height**. Group it under Dimensions, and make it a type parameter.
4. Add a label called **Angle Depth**. Group it under Dimensions, and make it a type parameter.



It is time now to make the actual angle sweep. Since you have a reference line and two parameters controlling the size, all you need to do is sketch the angle profile you wish, in Solid Form ➢ Solid Sweep.

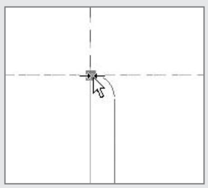
1. Open a 3D view.
2. In the Family tab, choose Solid Form ➢ Solid Sweep.

3. In the Sketch bar, click the Pick Path button.
4. Click the bottom reference line. A node and a reference box will appear at the midpoint. Click Finish Path.
5. Click the Profile button on the Sketch tab.
6. On the Options bar, click the Edit button.
7. In the Project Browser, open the Left elevation. This will allow you to sketch the angle in profile.
8. Draw your angle.



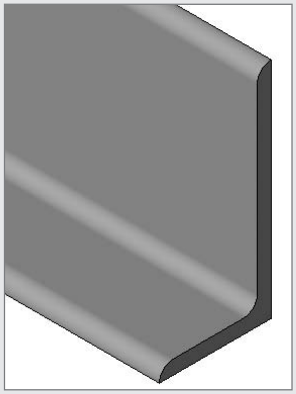
To ensure the angle will flex along with the reference planes, you will need to align and lock the endpoints to the reference planes. At first this can be somewhat daunting, but once you get the hang of it you can use this method in many places in Revit Structure.

1. Click the Align button.
2. For the first alignment point, select the top horizontal reference plane that represents the angle height.
3. For the second alignment point, select the endpoint of the magenta line representing the back of the angle. Once you select both points, a blue lock icon will appear. Be sure to click this icon, locking the alignment in place.



4. Because the radial portion needs to be locked too, you need to repeat the process for the endpoint of the arc. That means you basically have to align and lock two endpoints to the same reference plane.
5. Repeat the procedure for the leg of the angle where it intersects with the Angle Depth reference plane.
6. On the Sketch tab, click the Finish Profile button.
7. Again on the Sketch tab, click the Finish Sweep button.
8. Choose a 3D view.
9. Flex the angle by clicking the Family Types button and changing the angle size values.

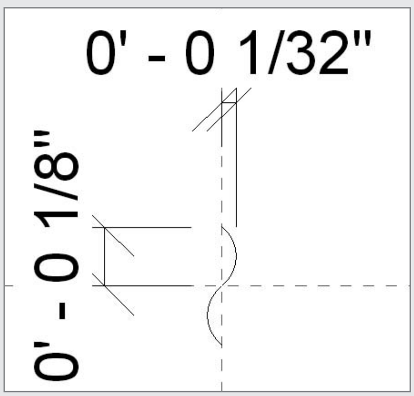
10. Save the file as `Joist Bridging.rfa`.



It is now time to work on the graphical plan representation. In plan view, the symbolic line work will look like the following graphic.

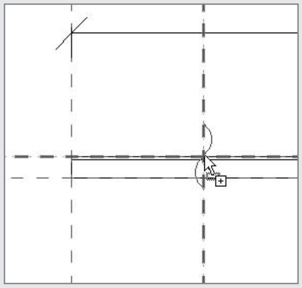
Of course this plan representation should also react to any scale change that may occur as well. To add the break marks at the end of the dashed lines, it would be prudent to actually make a separate annotation family and load it into this joist bridging family as a component.

1. Choose File ➤ New ➤ Annotation Symbol.
2. Select `Generic Annotation.rft` as the template.
3. Read the red block of text and then delete it.
4. On the Family tab, click the Lines button, and draw the two arcs as illustrated here. You can make the approximate dimensions the same ⅛″ × 1⁄32″.

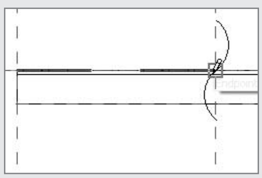


5. Save the file as `Tick Mark 1.rfa`.

6. Click the Load Into Projects button on the Family tab.
7. If you have more than one file open, select the `Joist Bridging.rfa` file.
8. In the `Joist Bridging.rfa` file, go to the Ref. Level plan view.
9. Click the Plane button on the Toolbar.
10. Set the level to Ref. Level in the dialog box, and then click OK.
11. Click the Symbol button on the Family tab.
12. Insert the Tick Mark 1 symbol at the intersections of the center reference plane and the symbolic offsets.



13. Click the Align button, and align and lock the symbol both vertically and horizontally to the reference planes. Remember, select the reference planes first, and then click the symbol's embedded references. When you cursor over the symbol, a center line will appear, allowing you to select the symbol. Of course, remember to lock the symbol.
14. Repeat steps 11 through 13 for the other side.
15. Open the Family Types dialog, and change the values for the symbolic offsets to test the movement of the newly added symbols.
16. On the Family tab, click the Symbolic Lines button, and then in the Type Selector menu on the Options bar, select Hidden Lines [Projection].
17. Draw a line from the intersections of the reference planes as shown in the graphic below (Although you do not need to in this case, you can align and lock the endpoints of these lines to the intersections).

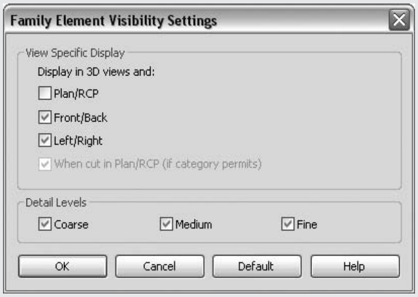


18. Draw another symbolic line at the other end.

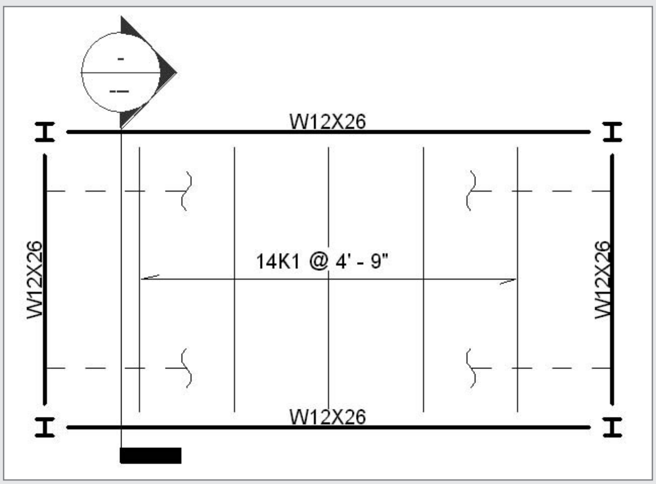
One last thing to do before you load this into the model and test it is to prevent the actual angle from showing up in plan view. Not only does this procedure enhance the functionality of the component, but it actually will decrease the amount of RAM being gobbled up while viewing the plan.

1. Select the angle (from any view is fine).
2. On the Options bar, click the Visibility button.

3. Under the View Specific Display field, uncheck Plan/RCP. If you want to, you can even specify which display level (Coarse, Medium, or Fine) the angle will show up in.



4. Click OK.
5. Choose Settings ➢ Family Category And Parameters.
6. Change the Family category to Structural Framing. (After you do this you may have to check to see if Revit Structure changed the symbolic lines to a continuous line type. If so, you can just change the setting back to Hidden Lines [Projection].)
7. Click OK.
8. Start a new project, and add some 14K joists (or find a project with joists already established).
9. On the Family tab, click the Load Into Projects button.
10. On the Modelling tab, click the Brace button.
11. Find the Joist Bridging family section.
12. Click the Element Properties button.
13. Change the Joist Size value to the appropriate size. (Notice that the other values that were part of a formula in the family are grayed out.)
14. Draw the bridging in plan view.



15. Cut a ¾″ x 1′-0″ section through the bridging. It should fall directly where you want it. The nice thing is you can move the angle into the correct position from the section, and it will update the placement in plan view as well.

That completes this exercise. How did you do? In this exercise you created an uplift/joist bridging family. The family works by your drawing it on a roof framing plan as a single line. As you have seen, the graphical result will be that in plan view there is symbolic line work representation. In section view there is an angle seated on the bottom of the joist for detailing purposes.

The final version of this family is available for download from the book's website at `www.sybex.com/go/masteringrevitstructure2009`. Also provided is the tick mark symbol used in the joist bridging family.

While creating a family externally from the model is a very common practice, there will be times where you still need to create a family but need surrounding geometry from the model to really build the family correctly. Or, if the family is so specific to that project that you know it will never be used again, you can create what is called an in-place family.

In-Place Families

In-place families can be a very powerful tool to help you in the design process. They can also be restrictive if they are created beyond their usefulness. Normally an in-place family is designed to be just what its title indicates. Although you can ultimately copy it around the model after it is created, an in-place family depends on specific placement within the model. That being said, if you intend to create an in-place family, be sure it is not going to be copied too much. If you find that you will need multiple instances of the family, create a family file (`.rfa`) and load it into the model. For example, Figure 18.26 shows two roofs and a special truss that was created using an in-place family. Because there will never be another project that would use these shapes, or any variation of them, it is beneficial to add an in-place family.

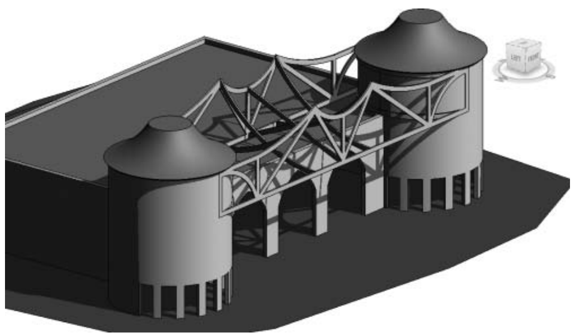


FIGURE 18.26
This kind of geometry lends itself well to the use of in-place families.

Also, an in-place family should be used in most situations where 3D massing is required. The way 3D massing works is that you must first add the masses and then use the faces of the masses to apply items such as roofs, walls, and floors. If you simply sketched an item such as a truss using only the 3D tools on the Massing tab of the Design bar, the item would not plot. Massing is simply used as a guide. When you sketch the same 3D truss as an in-place family, Revit Structure then knows that this is the final geometry.

To start with an in-place family, the first thing you need to do is go to the Modelling tab of the Design bar. At the bottom of the Modelling tab is a button that says Create, as shown in Figure 18.27.

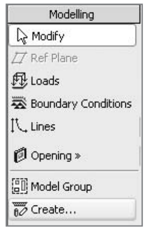


FIGURE 18.27
At the bottom of the Modelling tab of the Design bar you will see the Create button.

This is the button that allows you to create an in-place family. Once you click the button, a dialog box will open that lets you qualify the family. This is the same Family Types And Parameters dialog box you used to create an external family. In this example, you will add a truss to the sides of the model. You will select Structural Framing from this dialog box. Once you choose the Family Type value, the next dialog box allows you to name the specific family. This is less important than specifying the actual family type, but you should always adhere to good naming conventions. For the example, the name will be Esthetic Trusses. Once you are finished naming the new family, Revit Structure will change the Design bar to contain only the Family tab, as you saw in the Family Editor earlier in the chapter. This is done to give you the full capability to create a family just as you would in the external editor, only this time you can use the existing geometry in the model as a guide. The best of both worlds! Also, you still have access to the Project Browser to navigate to the different views.

Usually, creating an in-place family as opposed to creating an external family will be less difficult based on the fact that this family does not need to be nearly as dynamic as an inserted family. The reason is the dimensions are fixed and the nature of the placement is more static. There will be fewer, if any, variables. In the case of adding a specific item to the side of a building, the only reference planes you may need are the walls, floors, and roofs of the actual structure.

Since you are in the Family Editor mode, it is now safe to add as much 3D geometry as you wish. As we mentioned earlier, however, once you start adding 3D massing, Revit Structure will need some reference. For example, to create a truss profile, on the Family tab of the Design bar choose Solid Form ➢ Solid Extrusion. Instead of simply letting you place the extrusion in the model, Revit Structure needs to know where on the building the extrusion will be placed. It makes sense. Revit Structure has no idea whether you want the geometry to sit on a wall or to be

100 miles back in the next county. When it prompts you for a work plane, Revit Structure allows you to pick something related to the building to establish the plane, as shown in Figure 18.28.

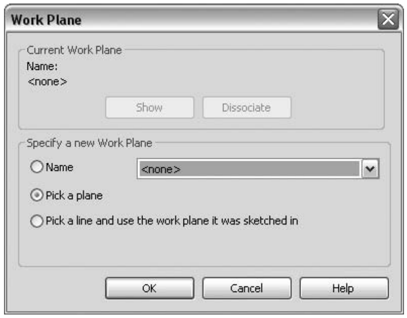


FIGURE 18.28
The Work Plane dialog box allows you to specify the location of the building where your family will reside. It is common to choose a wall or a framing member to establish a position.

In this example, the entire face of a wall is being selected as the work plane in which to start modeling the special truss, as illustrated in Figure 18.29. You can use any item as a reference, and you can also use established gridlines.

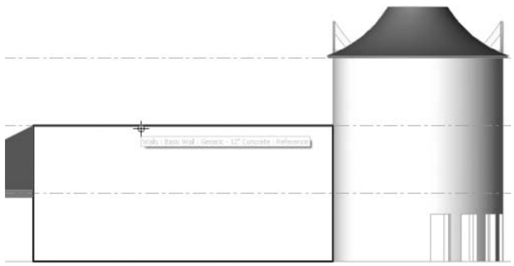


FIGURE 18.29
Select the surface of a wall on which to start modeling the new family.

Now you are essentially free to sketch anything you want. If you add reference planes, they will disappear when you finish the family. If you choose to come back and alter the family at a later date, the reference planes will reappear, as they are embedded in the family itself. Figure 18.30 illustrates an in-place family in production.

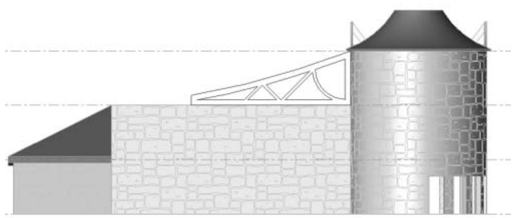


FIGURE 18.30
This is an extrusion within an in-place family. Notice that you can basically draw any shape you need with the same ease as you had when drafting.

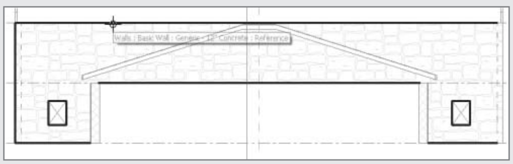
Once you have finished with the 3D solid, you can choose Finish Sketch on the Family tab. Notice that when you do that, you are still in the Family Editor. This means you can perform more 3D functions. In addition, if you look at the Family tab, you will see that you can also insert components and detail components into the family.

To investigate further into the creation of an in-place family, you can open the Revit Structure file called `Gothic Structure.rvt` and follow along with the next exercise.

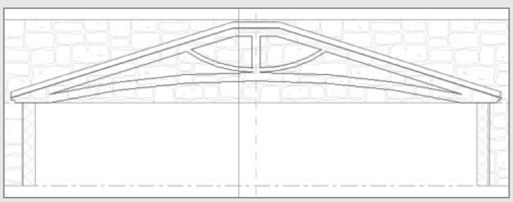
Exercise: Creating an In-Place Family

This lesson will walk through the creation of a special truss family designed specifically for one building that will never be used again in another project.

1. Open the file, available at the book's website, called `Gothic Structure.rvt`.
2. The file should open by default in the section view called Truss Section - 1. If not, open the view called Truss Section - 1.
3. On the Design bar, go to the Modelling tab and click the Create button.
4. For the Family Category value, select Structural Framing and then click OK.
5. For the name, enter **Back Entry Truss**, and click OK.
6. On the Family tab of the Design bar, choose Solid Form ➢ Solid Extrusion.
7. You will encounter the Work Plane dialog box. Under Specify New Plane, choose Pick a Plane, and click OK.
8. Select the back wall.

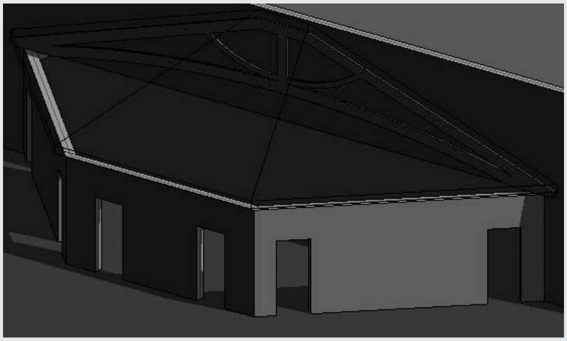


9. Now you are ready to start sketching. Remember, the 3D extrusion will extrude toward you.
10. Sketch a truss shape, complex or simple—it doesn't matter.



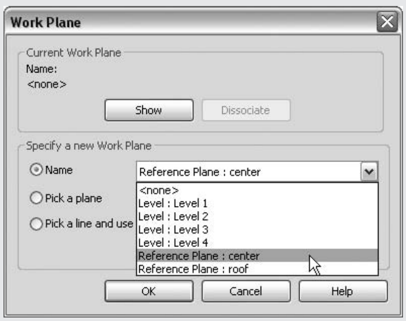
11. Click the Extrusion Properties button on the Sketch tab.
12. In the Element Properties dialog, make sure the material selected is Wood – Birch.
13. Set the Extrusion End value to **3″**.

14. Click OK.
15. On the Sketch tab, click Finish Sketch.
16. On the Family tab, click Finish Family.
17. Go to the default 3D view, and spin around the point of view to check out your truss.

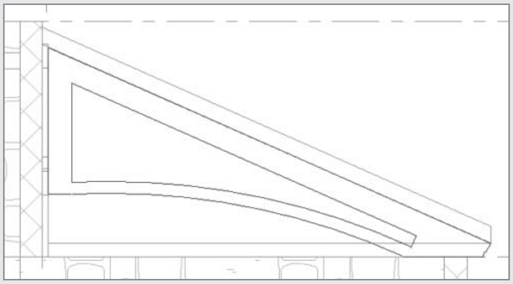


Now you will display Truss Section - 2. This will bring you to the lateral view of the truss. Just because you finished the family, that does not mean you cannot open another view and continue building the family.

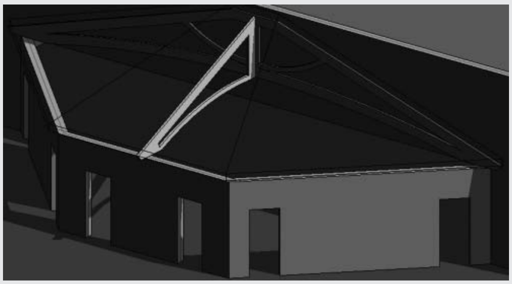
1. Open the section view Truss Section - 2.
2. Select the truss you previously created, which is against the building.
3. On the Options bar, click the Edit button. This brings you back into the Family Editor.
4. Click the Solid Form ➢ Solid Extrusion button.
5. When prompted to pick a plane, select Reference Plane : Center from the drop-down list.



6. Sketch another truss profile.



7. Click the Extrusion Properties button.
8. Set the End value to **1½″**.
9. Set the Start value to **-1½″** (negative).
10. Click OK.
11. Click Finish Sketch.
12. Click Finish Family.
13. Check out the truss in your 3D view.



The ability to freely create any kind of geometry necessary in a 2D environment and achieve 3D results is crucial to BIM and the whole modeling process. As you can see from the previous exercise, all you need to do to elaborate on the truss system being created is to add perhaps two sections along the hips and sketch the trusses in those views. Because is the additions are still part of the same family, you can experience greater organization and flexibility if you need to alter the trusses for whatever reason.

Another item in Revit Structure that allows for organization is the Group feature. The ability to take any typical geometric configuration and create a unified set of objects from it can be quite advantageous.

Grouping

The concept of grouping has been around in Revit Structure for quite some time, and it is a simple procedure. You collect a bunch of components, group them together, and copy the entire group around the model. If you decide you need to change the grouping configuration, you change one, and the rest are updated. It's as simple as that. Of course Revit Structure 2009 would not best Revit Structure 2008 without adding a few enhancements to groups, allowing you more flexibility and control over the behavior of the group.

Creating a Group

Before you create a group you need to first ask yourself, "Should this be a group?" Like anything in Revit Structure, if you use a specific feature beyond its usefulness, you are negating the benefit of that feature's functionality. You can wind up with a collection of groups that get confusing and, worse yet, will exponentially expand your file size. So with that in mind, let's start creating groups. In our example, a typical cross-bracing elevation will be grouped and then labeled accordingly.

The purpose of a group is to maintain consistency throughout the model by establishing typical conditions and configurations. The end goal, as is the overall end goal of BIM, is to ensure that a change made in one place will occur in every instance of that configuration. There will be times, however, when a copy of a specific group needs to have a stand-alone edit performed. Revit Structure takes this into consideration as well. But first, take a look at Figure 18.31. It is an elevation of two slightly different bracing schemes: Brace Frame A and Brace Frame B. It sure would be nice if these could be grouped and copied around the model. It would be even better if when the HSS sizes change, every instance of Brace Frame A and Brace Frame B would be updated.

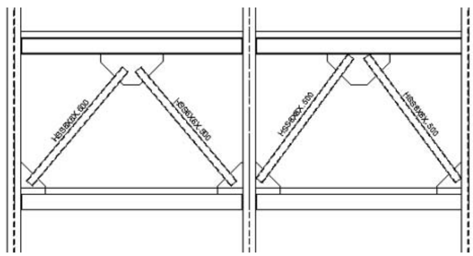


FIGURE 18.31
These brace frames are stand-alone right now, but they will be made into a group later.

To create the group containing the bracing, you must first select the items you want grouped. This includes any text and labels. You then click the Group button on the Edit toolbar, as shown in Figure 18.32. In this example, the brace frame to the left is selected first. This will become a group called Brace Frame – A.

FIGURE 18.32
You first select the items you want grouped, and then you click the Group button on the Edit toolbar.

Once you click the Group button, a dialog box will appear, allowing you to label two separate items. One item is Model Group, which contains the actual components that compose the grouping. Name the model group **Brace Frame - A**. The second name is for any text or labels that go along with the components. If you have a group with no annotation, this second naming step is omitted. For this example, name the Attached Detail Group item **Brace Frame - A Labels**, as shown in Figure 18.33.

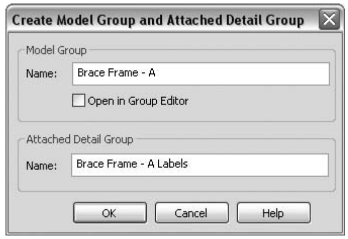


Figure 18.33
The Create Group dialog box allows you to label the model geometry as well as specify the attached detail (annotation).

Once the group is created, you will see a blue grip at the bottom center of the group. This is the group origin, and it can be moved. It is basically an insertion point for the origin of the group. Make sure this origin is exactly where you want it. See Figure 18.34.

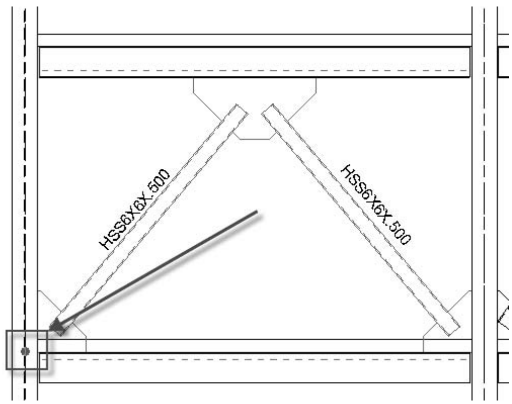


Figure 18.34
Drag the origin to the point that makes the most sense as an insertion point.

Once the group has been established, it will appear in the Project Browser. Because it is a grouping of model objects, it will appear under the heading of Model. If you expand the model tree, it will expose the Brace Frame - A group. Now if you expand the Brace Frame - A group, you will see the attached detail group, indicated by the paperclip icon, as shown in Figure 18.35.

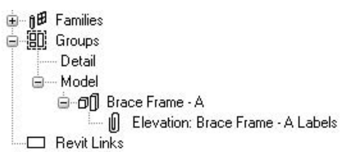


Figure 18.35
The group is displayed in the Project Browser. You can simply drag and drop the group into the model from here.

Now that you can see the group in the Project Browser, you can simply drag the group into the model and place it at the insertion point. When you are selecting the group item to drag into the model, be sure to drag the actual model group, not the attached detail. Once the actual model group is placed into position, you will notice that the labels did not move along with it. This is because you may not want to actually include all of the text and tags in every occurrence of the group. In many cases, you can label one typical group and assume the others are the same unless noted otherwise. This is okay though, because you can always label the entire group in one shot.

Once the group is positioned, select it. You will see that the entire group is one entity with a blue dotted line indicating the perimeter of the group. On the Options bar there are four choices: Edit Group, Ungroup, Place Detail, and Link. The first choice, Edit Group, allows you to make changes to the group's configuration. Once you click the Edit Group button, you will be thrown into the Group Editor, as shown in Figure 18.36.

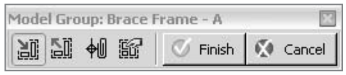


Figure 18.36
In edit mode, you can add any geometry to the group, remove geometry, and edit the geometry that composes the group. Once you have finished, you can close out, and all of the occurrences of that group will be updated throughout the model.

Ungroup

The Ungroup button obviously removes the selected group from the grouping and returns the geometry to its original state. In a sense, it explodes the group.

Place Detail

The Place Detail button will add the labels and text that compose the Attached Detail group. This is a nice feature because it labels the members consistently. Once you click the Place Detail button, a dialog box will appear listing the attached details related to that group. You can select a detail, and it will label the entire group, as shown in Figure 18.37.

Link

By clicking the Link button, you are actually exporting the group to a separate Revit Structure model file and then linking it back into the model. Once you click this button, Revit Structure will ask you if you want to create a new model or if you wish to find an existing model, as shown in Figure 18.38.

FIGURE 18.37
This dialog box appears giving you the choice of which attached detail to include. You access this dialog box by selecting the group and then clicking the Place Detail button on the Options bar.

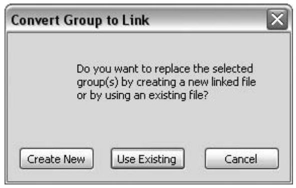


FIGURE 18.38
You can create a linked Revit Structure model by simply selecting any group and clicking the Link button on the Options bar. This can be advantageous in keeping the active Revit Structure file size to a minimum.

Normally you will create a new model, but there are some cases such as an architectural overlay where you may want to find an existing model. Once you choose between creating a new model and linking an existing one, Revit Structure will add the link to the Project Browser, as shown in Figure 18.39. You can now work on the link separately or have another team member work on the model without having to start file sharing. This can become quite helpful in the waning days of the project.

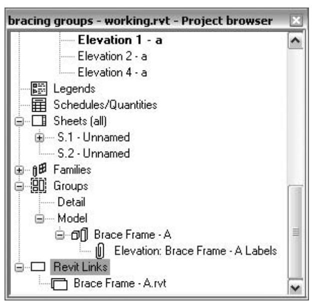


FIGURE 18.39
The new link is added to the Project Browser.

Another nice thing about the ability to turn a group into a linked Revit Structure model is that you can reverse the action. If you select the new link, you will notice a Bind button on the Options bar. You can bind any linked model (it does not have to start as a group). Once the link is bound, it turns into a group. This can be helpful once the project goes out the door, and the next day you can bind the project back together. Or, if you would like to physically include an architectural model or an MEP model, you can bind the linked model and basically merge it into your structural model.

Once you click the Bind button, you will get a dialog box asking which items you want to bring into the current model. If you choose to bring in levels and grids, you will create separate, uniquely named grids and levels. Be careful here, as this could overwhelm you with redundant information. See Figure 18.40.

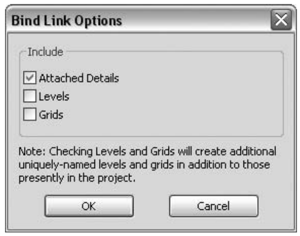


FIGURE 18.40
Revit Structure gives you the choice to not add redundant data when you are converting a link to a group.

EDITING GROUPS INDEPENDENTLY

You will almost certainly run into a situation where you would like to make a slight modification to only one instance of a group without actually having to ungroup the set and then create a brand-new grouping. This is allowable in Revit Structure. Once the group has been copied around, you can simply hover your cursor over the object you wish to modify within the group. Of course, the entire group will become highlighted. Once this occurs, press the Tab key on your keyboard. Now only that specific element is highlighted. Select the item, and it becomes selected with a small group icon appearing, as shown in Figure 18.41. You are given the choice to either exclude the item from this copy of the group or to physically take the item and move it out of the group and back into the project.

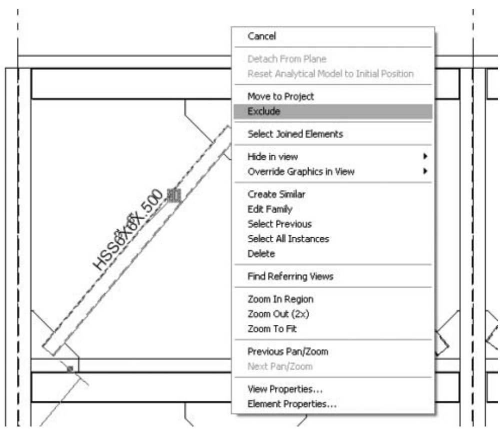


FIGURE 18.41
By selecting only the single member, you can either move it to the project, thus removing it from the group, or you can exclude it altogether.

If you choose to exclude the member from the group, Revit Structure remembers there was an instance of that member for later retrieval. After you select Remove, you can hover your cursor back over the group. When the group highlights, the member you removed reappears. You can then hover the cursor over the group icon that appears and click it to move the member back into the group, as shown in Figure 18.42.

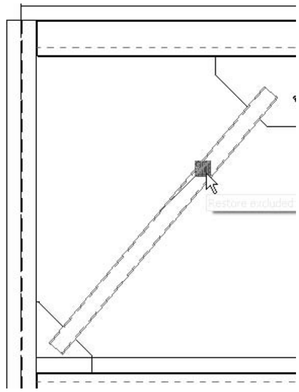


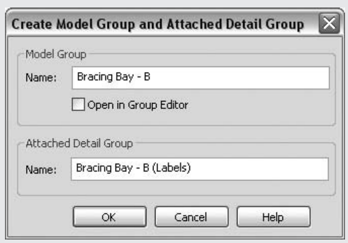
FIGURE 18.42
By hovering your cursor above a group with an excluded element, you can select the group icon on the member that was removed to return it to its original state within the group.

The following exercise will allow you to create a group on your own using the provided Revit Structure model on the book's website.

EXERCISE: CREATING A BRACE FRAME GROUP

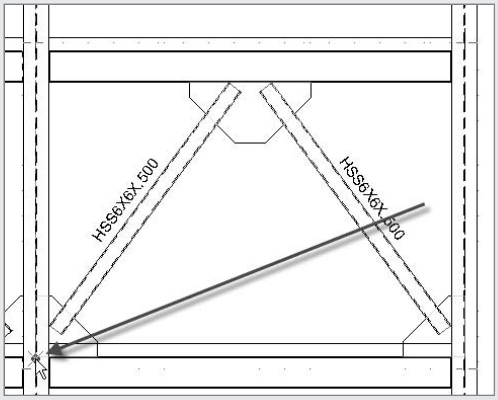
This exercise allows you to create a brace frame group and copy it throughout the model. As you are running through this exercise, try to think about specific situations where you may want to use this feature.

1. Open the file titled `Brace Frame - A Labels`.
2. Select the bracing element to the right. This includes the HSS members, the three plates, and the labels. Be careful not to select the adjacent framing or the slab.
3. Click the Group button.
4. Name the Model group **Bracing Bay - B**.
5. Name the Attached Detail group **Bracing Bay - B (Labels)**.

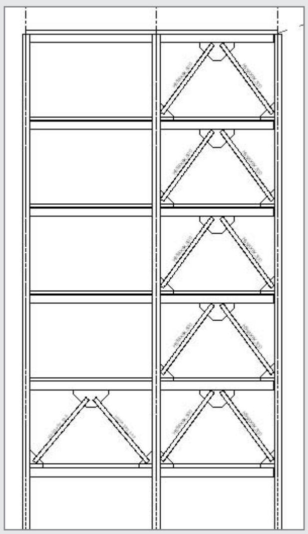


6. Click OK.

7. Find the grip dot that indicates the group's insert point of origin, and drag it to the left projected end of the HSS framing.



8. Expand the Project Browser under the Groups category, and find the new group.
9. Drag the new group into the bay above.
10. Select the new group.
11. Click the Place Detail button on the Options bar.
12. Click the Elevation: Bracing Bay - B (Labels) check box.
13. You can either repeat the procedure, or you can simply copy the bracing to each separate bay. Either way, fill in the rest of the bays above Bracing Bay - B.



14. Select any one of the brace groups, and click Edit Group on the Options bar.
15. Select the two HSS6X6X.500 posts, and change them to W8x10s from the Type Selector on the Options bar.
16. Click Finish on the Temporary Model Group toolbar.
17. All of the braces are then updated.

As you can see, there are some real advantages to using groups within a Revit Structure model. If you plan out a good grouping scheme that makes sense to everyone on the team, you will greatly increase your productivity in terms of organization and uniformity.

The Bottom Line

Create a footing step family. Creating families is a vital skill. You create a family, such as a footing step family, by meticulously tying reference planes with dimensions and parameters. Then you add 3D solids to the references to achieve a flexible, useful family.

Master It When you start a new Revit Structure family from a template, there will be existing reference planes. How do these reference planes help in the creation of the family?

Create in-place families. You create specialized families, called in-place families, directly within the model, tying the family to the surrounding building. This allows you to more easily create custom geometry that will probably never be used in another model.

Master It What is the process for creating a custom family directly within the model?

Create groups. Adding groups to the model greatly reduces the time spent organizing and manipulating the configuration of certain items. Also, by linking a group, you can actually create a separate Revit Structure model and link it back into Revit Structure similarly to create an x-ref in AutoCAD.

Master It You also learned that a linked Revit Structure file can be turned into an embedded group. Explain the procedure for this to occur.

Chapter 19

Advanced Structural Families

As you have been working through the chapters of this book, you have been learning how to use the basic libraries to obtain your model elements. You go to a library file and load in different types from a family for use in your project, such as a 3 × 6 from the lumber family. But as you gain experience and have to deal with the many objects that occur in a real-world scenario, you quickly find that the built-in libraries do not have everything you need. You'll get to more complex objects such as wood and steel trusses, tapered steel girders, and wood beam nailers, to name just a few. These objects will require more advanced modeling techniques.

This chapter will look at some of these objects that you will undoubtedly encounter and need to model and will look for the more advanced methods to construct them. Integral to this discussion is the notion of pure, solid modeling techniques that enable you to construct modeled objects. Using a combination of solid and void forms that can be extruded, revolved, swept, and blended, you will find that you can create almost any shape imaginable. These tools are located on the Modelling menu and accessed through the Create option. In the examples presented in this chapter you will learn to use these tools to construct particular families for your project. When you have completed this chapter, you will have learned how to create more complex objects.

That is a tall order for one chapter, but you really do want to move beyond the simple techniques in order to unleash the great power of this program. We will break the processes down into easy-to-follow steps that you can then apply to your own families when the need arises. The examples that you will learn in this chapter are not perfect, nor are they the final word on how a technique might be done. Perhaps if you tinker a bit, you can improve on them. The first object you will learn how to construct is a tapered girder family.

In this chapter you will learn to:

- Create a parametrically driven tapered steel girder family
- Construct an in-place bent beam family
- Adapt the steel wide flange beam family by adding a nailer to its top
- Create an elevator pit family that can be inserted into your project
- Produce wood and steel truss families using the truss template

Tapered Steel Girders

Tapered steel girders are a common element in many buildings but are not available in any Revit Structure family (see Figure 19.1). You will need to prepare your own at some point. Many long span conditions, such as a gymnasium or a hotel lobby, might use this type of girder. A tapered steel girder is one whose center depth is greater than the end depth, or it is flipped

upward with the center higher than the endpoints. You could make a single nonparametric tapered girder specifically for your project by making an in-place family (it is nonparametric if it does not flex into different depths). But why not create a family that you can use over and over again and from which you can easily create different mid-span and end-span depths? That is the most efficient approach, though a little more thought provoking and challenging to create. When you achieve this level of understanding of how to develop from scratch a particular family, you begin to really unlock the power of Revit Structure.

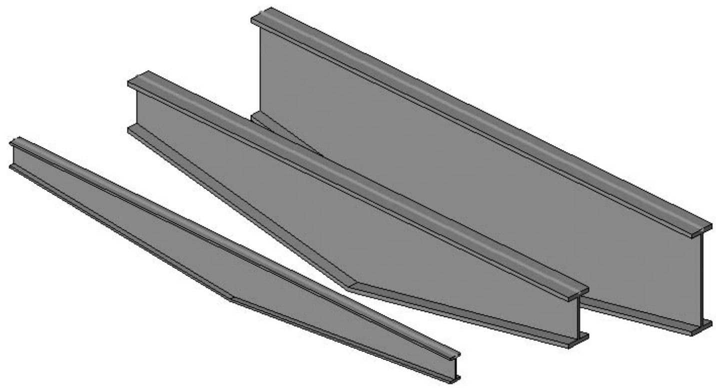


FIGURE 19.1 Tapered steel girder types created from a parametric family

Create the Basic Constraints

To get the process started, first you will create a new family, and then you will define the basic constraints of the shape. As you learned in the previous chapter on basic family creation this is done by adding reference planes to which you will lock the girder sketch lines that define its shape. Those reference planes will be dimensioned, and then the dimensions will be enhanced so that they become labels whose value may vary. That will allow the shape to flex as you create different types.

In the next step you will create the shape by using a sweep blend to create the taper form, which will be attached to the reference planes. Finally you will adjust the visibility of the various family components so that they show correctly in coarse or 3D mode.

The first step in making a parametric tapered steel girder family is to open a new family template file and set up some reference planes, so do the following:

1. Open a new project.
2. Start a new family by choosing File ➤ New ➤ Family on the menu bar.
3. Select Structural Framing - Beams and Braces for the template, and then click OK.
4. Once you're in the Family Editor, display the front elevation.
5. Delete the rectangular extrusion that you find there.
6. Change the display mode to Medium so the stick symbol will be hidden.

Now you will need to add five horizontal reference planes: one for the top flange of the girder, two for the end-span depth, and two that represent the mid-span girder depth (see Figure 19.2).

7. On the Family tab click Ref Plane, and draw the five new horizontal reference planes:
   A. Draw one just below Ref. Level (about 1″ down).
   B. Then draw two (about 1″ apart) about 1′ below Ref. Level.
   C. Then draw another two about 2′ below Ref. Level.

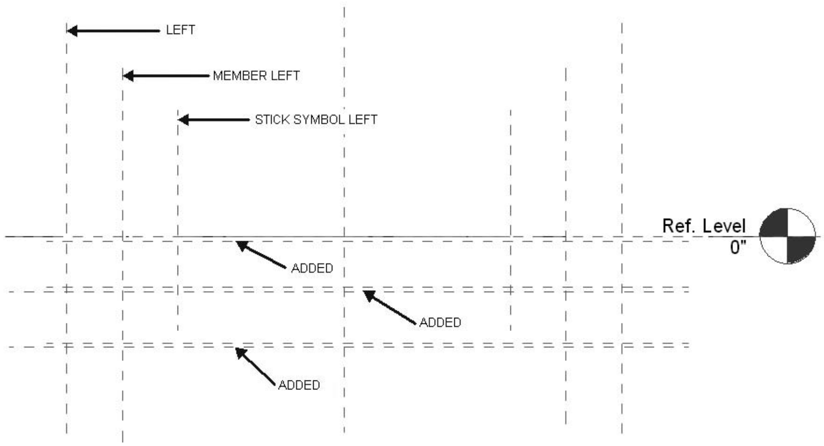


FIGURE 19.2
The front elevation with five added reference planes

Next go to the right view so that you can add four more vertical reference planes that will represent the flange width and the web thickness.

8. On the Family tab click Ref Plane, and draw four new vertical reference planes like those in Figure 19.3 (note that the girder profile has been ghosted in for the next few figures to make it easier for you to understand what we are doing). Draw two reference planes, at ½″ and 3″, to the left of the center reference line and two to the right of it.

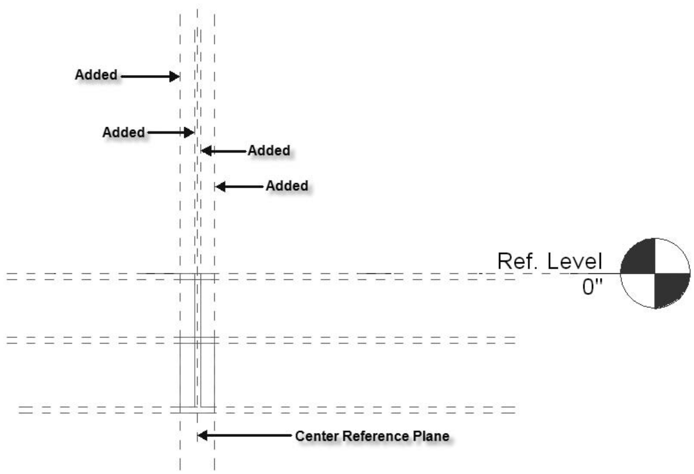


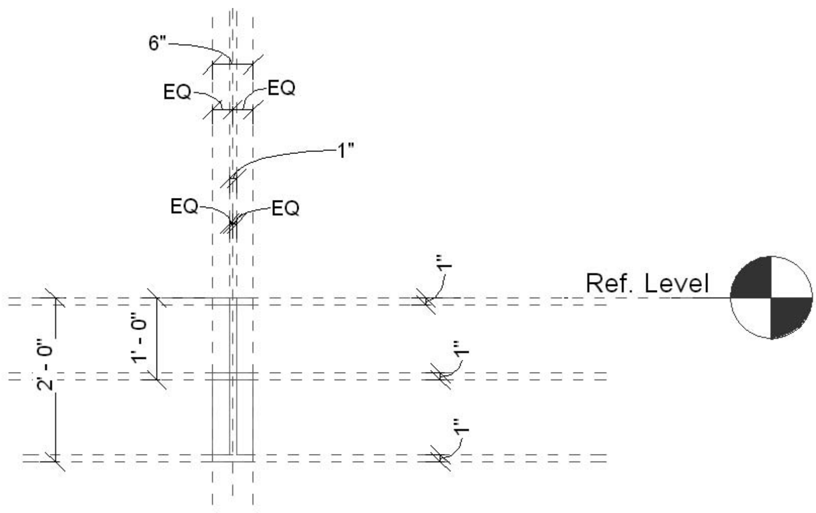
FIGURE 19.3
Adding the reference planes to the right side view

Adding Dimensions and Labels to the Constraints

Now that the new reference planes have been added, you will dimension them and apply labels. Your values do not have to match the figure at this time. To dimension the reference planes do the following:

1. In the right view add seven defining dimensions to the reference lines (see Figure 19.4) for the following:
   - **A.** End depth
   - **B.** Mid depth
   - **C.** Flange thickness in three locations
   - **D.** Flange width (You must also add EQ dimensions to keep the flange symmetrical about the centerline.)
   - **E.** Web thickness (You must also add EQ dimensions to keep the flange symmetrical about the centerline.)

Figure 19.4
Adding dimensions to the right side view



Now that the reference planes are dimensioned, you will create the labels. You want to make the family parametric so that you can create different family types in your projects. To do this you will alter the dimensions you just added in order to create labels for end-span depth, mid-span depth, flange width and flange thickness, and web width and web thickness.

1. Select the End-Span Depth dimension (1´-0˝), right-click, and select Edit Label from the context menu.
2. Choose <Add Parameter>.

3. In the Parameter Properties dialog box name the parameter **End-Span Depth**.
4. Choose Dimensions as the group parameter.
5. Click the Type radio button to make it a type parameter.
6. Click OK to complete the label (see Figure 19.5).

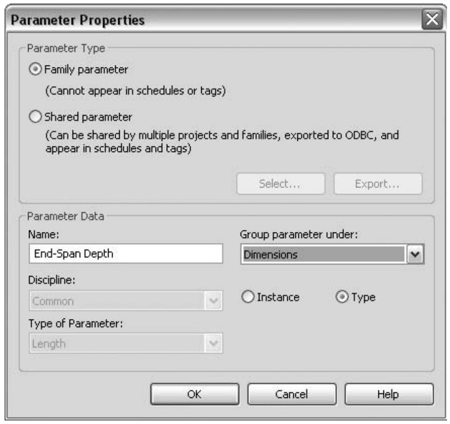


FIGURE 19.5 Defining the End-Span Depth label in the Parameter Properties dialog box

7. Repeat steps 1 through 6, and create the Mid-Span Depth, the Flange Width and Flange Thickness, and the Web Thickness parameters. When completed, your reference planes should look like Figure 19.6.

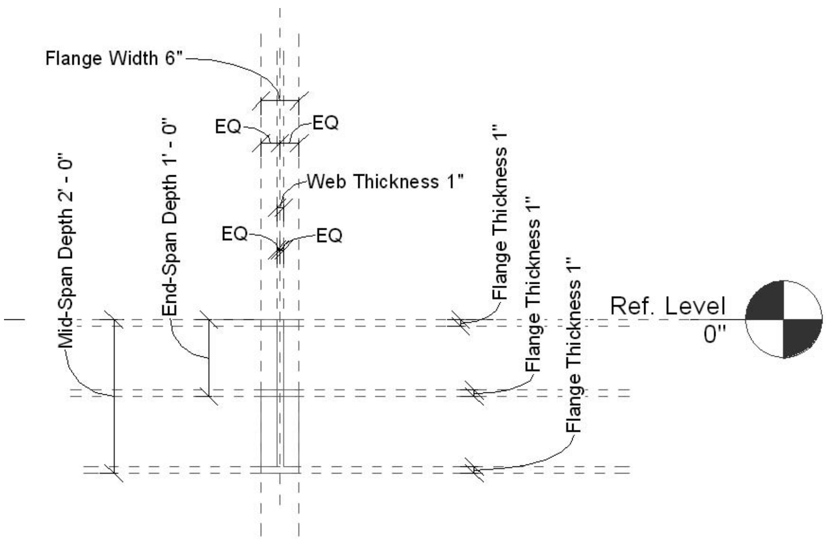


FIGURE 19.6 Altering the dimensions to create the labels

Now you need to start flexing the labels to test whether the reference lines you have created are working. Remember that you need to constantly test your family by flexing the different dimensions you have established.

1. Click Family Types on the Family tab.
2. Change the Mid-Span Depth parameter to **5′-0″** and then click Apply.

Did the bottom two reference lines stretch correctly to the new depth? If not, you need to cancel and check the problem areas. This process can get frustrating, but after awhile you will get the hang of it. Test the other labels in the same way until all are flexing correctly. Once you have the reference lines flexing correctly, open the Ref. Level plan view again.

Creating the Tapered Girder Geometry

You now will create the basic form and geometry for the tapered girder (look back at Figure 19.1). Once you do that, you are almost finished with this family creation. This part of the process will use a solid sweep blend to create half of the tapered girder.

1. Choose the Ref. Level plan.
2. On the Family tab choose Solid Form ➢ Solid Swept Blend.
3. On the Sketch tab click Sketch 2D Path.
4. Sketch a horizontal line, along the center x-x axis, from the left (leftmost) vertical reference line to the center reference line, and lock its ends to them.
5. Click Finish Path.
6. On the Sketch tab choose Profile 1, and then choose the right elevation view.
7. On the Options bar click Edit.
8. Sketch lines that represent the end-depth profile (see Figure 19.7).

It is very important to make sure the sketch lines are locked to the reference planes. To do so, follow these steps:

1. On the Tools toolbar select the Align feature.
2. Click a reference line and then the associated sketch line you want to match. Close the lock.
3. Do this for all the sketch lines you have created for the end-depth profile.
4. Click Finish Profile.
5. On the Sketch menu choose Profile 2.
6. On the Options bar click Edit.
7. Sketch lines that represent the mid-depth profile and lock them to the reference planes as you did with the end-depth profile.
8. Click Finish Profile, and then click Finish Swept Blend.

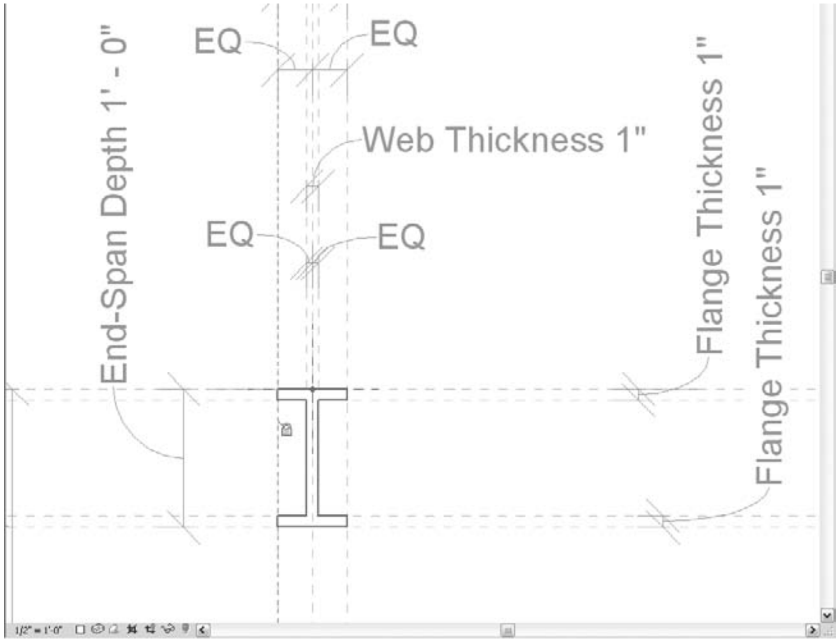


FIGURE 19.7
Lock the sketch lines to the appropriate reference plane.

Now go to a 3D view and flex the various values in the Family Types dialog box to test that the shape is stretching as you expect. When you are satisfied, go to the front elevation view.

1. Highlight the swept blend, and mirror it about the vertical center line.
2. Lock the ends of both swept blends to the vertical center line (see Figure 19.8).
3. On the Tools toolbar click the Join Geometry button, and join the two pieces together.
4. Lock the ends of the tapered girder to the left and right vertical reference planes.

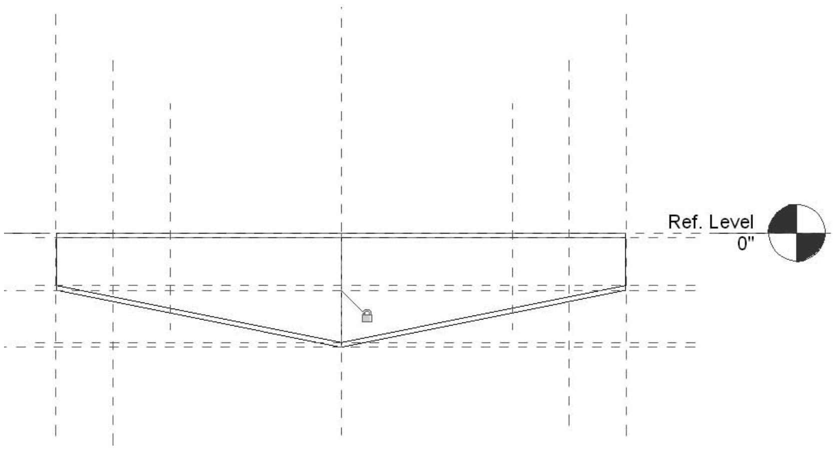


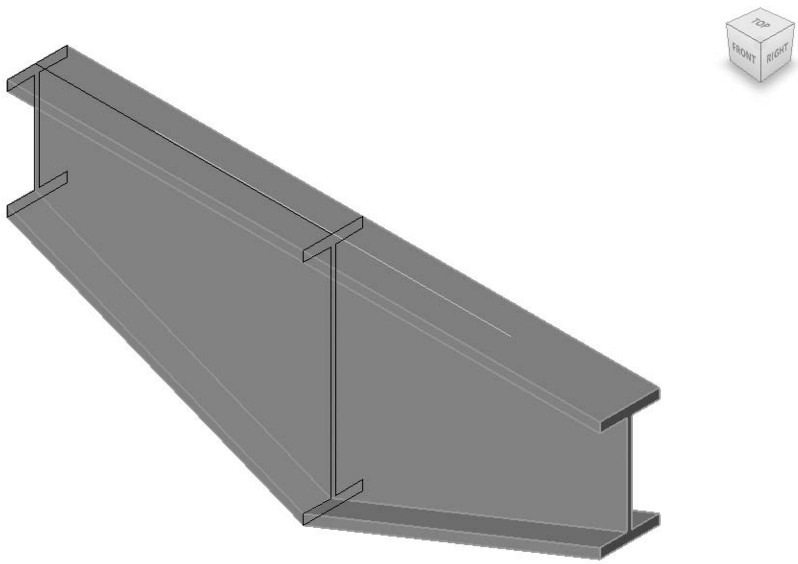
FIGURE 19.8
Locking the two segments of the tapered girder to the center line

It is very important to make sure the sketch lines of the new mirrored solid are locked to the reference planes as well. To do so, use the Align tool. This can get a bit tedious, but pay close attention so that you get all of the lines:

1. Adjust the family type End and Mid-depth values again, this time using **8´** for Mid and **6´** for End. You will now notice that the new right sweep isn't locked properly.
2. Select the right sweep, and then click the Edit Swept Blend button on the Options bar.
3. Now go to the right view. Select the lower profile, and then click Edit on the Options bar.
4. On the Tools toolbar select Align, and click a reference line and then the associated sketch line. Close the lock.
5. Do this for all the end-depth and mid-depth sketch lines for the mirrored profile, both horizontal and vertical edges.
6. Click Finish Sketch, and then select the upper profile and repeat steps 3 through 5. Once this is complete, click Finish Swept Blend. You can now return to the front view and flex your model to verify your locks.

You have now completed the tapered steel girder family (see Figure 19.9) and can now make some family types.

FIGURE 19.9
The two profiles of the solid swept blend highlighted for half the tapered girder



1. Choose Family Types on the Family tab.
2. Then in the Family Types dialog box choose New ➤ Duplicate.
3. Name the type **1´ × 2´**.

4. Change the End-Span Depth value to **1′-0″** and the Mid-Depth value to **2′-0″**.
5. Click Apply, and confirm that it flexed okay.
6. Repeat steps 1 through 3, and create a 2′ × 4′ family type and a 3′ × 5′ family type.
7. Save the finished tapered girder.

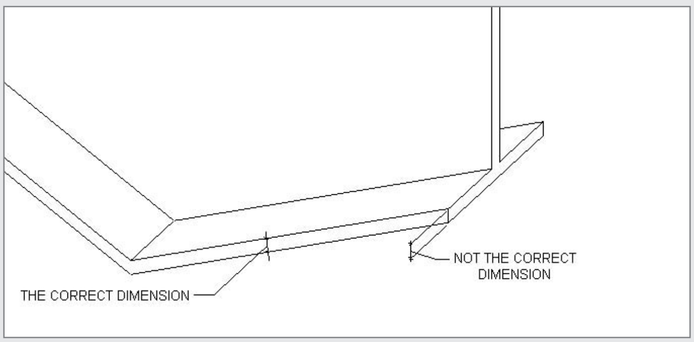
Now you are ready to test the finished tapered girder. It acts like any other beam object, and it also will work with a beam system.

1. Load the family into a given project.
2. Click Beam on the Modelling tab.
3. Choose a tapered girder type, and draw it just as you would any other beam.

Real World Scenario

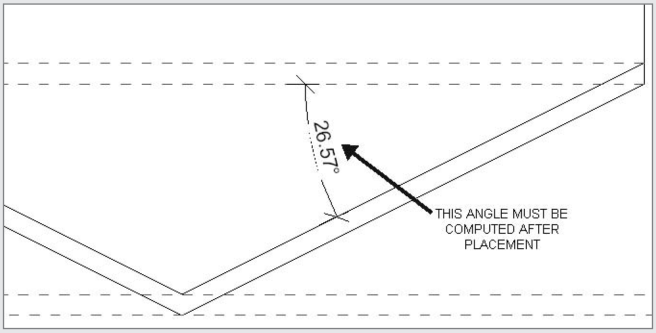
Oh-Oh! We Have a Major Problem Here!

Did you notice one discrepancy in what we have done? Remember how you set the flange thickness dimension and label in the right side view? But in fact the lower flange is sloping. That means the dimension in the right side view is not really the thickness of the lower flange but is a little bigger, as illustrated in the following graphic.



This turns out to be the most difficult and advanced part of creating this family. If you go to the front view and try to add a dimension and label for the sloping bottom flange, you will find that it becomes overconstrained. There is no way around it. So how can we overcome this dilemma? This problem even flummoxed the authors, who then had to contact Revit Structure headquarters for assistance. Coming to our aid was Jack Zhang Lee, one of the family-creation specialists.

To set the correct flange thickness for the sloped parts of the girder, we need to set the sloped angle beforehand (see the following illustration), so that we can then calculate the flange thickness with that angle. The angle for the tapered girder is a function of the overall length of the girder, so it can be computed only as a resultant value after placing the element. So you must test this family for accuracy in a project, not in the family file.



Yes, you can add formulas to your parameters for just this kind of purpose. Here is how it works.

In the family: You create a new parameter, namely, **Length1**, and set Length1 equal to **Cut Length**.

In the Family Editor, Length1 may not represent the real physical length of the beam, but in the project it does refer to the real physical length of the component. The flange thickness could be calculated precisely in the project with the nested profile employed.

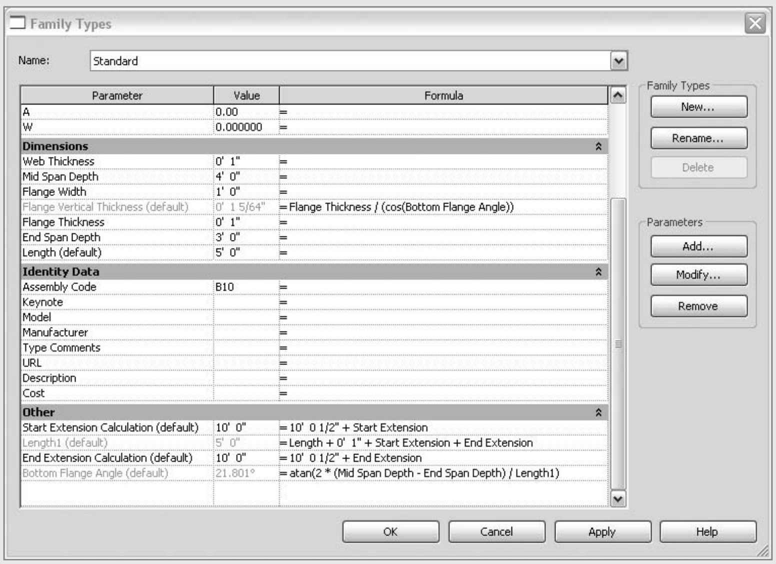
In the project: The resultant angle is computed with the following formula (see the two illustrations):

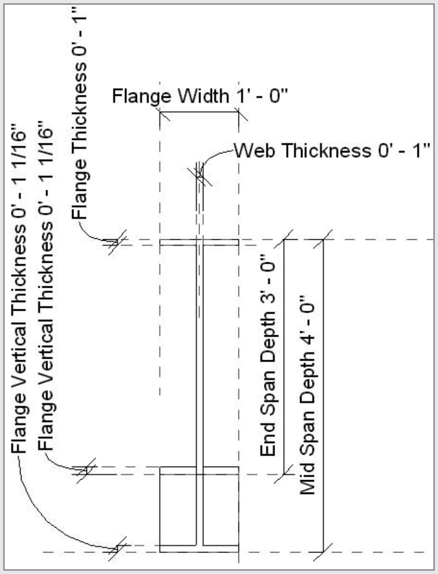
Bottom Flange Angle = atan(2 *(Mid Span Depth - End Span Depth) / Length1)

For the beam itself the cut length is computed with the following formula:

Length1 = Length + 0´ 1˝ + Start Extension + End Extension

Then from that the Flange Vertical Thickness value is computed. Notice how the three parameters are grayed out in the dialog box. That is because they are resultant values that you cannot manually change.





So there you have the answer to a very complex problem. I am sure we could go further in this discussion and keep tweaking the family, but we need to move on to our next advanced structural family. Maybe you can take a crack at it. You may find a better solution. When you get stuck with this sort of thing, it can be very taxing on your sanity! But stick with it, and look around on the Web for help in the AUGI Structural Forum or other such sites. Don't be afraid to ask for help. That's what we did.

Real World Scenario

The Old 80-20 Rule

You may know the old saying usually referred to as the 80-20 rule. It has many applications. For instance you probably wear only 20 percent of your clothes, while the other 80 percent just take up space in your closet. In the case of modeling, you will find that as you gain proficiency, 80 percent of the work will sail along. The client and your boss will be delighted that you have done the work and only used 20 percent of your drafting fee for the project. Life is wonderful at that point; high fives all around.

But then you find you have to model elements that cannot be done so easily and require huge work-arounds or are so complex that it takes a monumental effort to complete them. Suddenly that 20 percent of the work takes 80 percent of your fee to accomplish. The boss is probably not so happy at this point.

That is one of the reasons why we are presenting these more complex families to you in this chapter. These are objects that can take a lot of time to do if you do not create families for them. Creating them anew each time you need them will considerably slow your progress and create needless work.

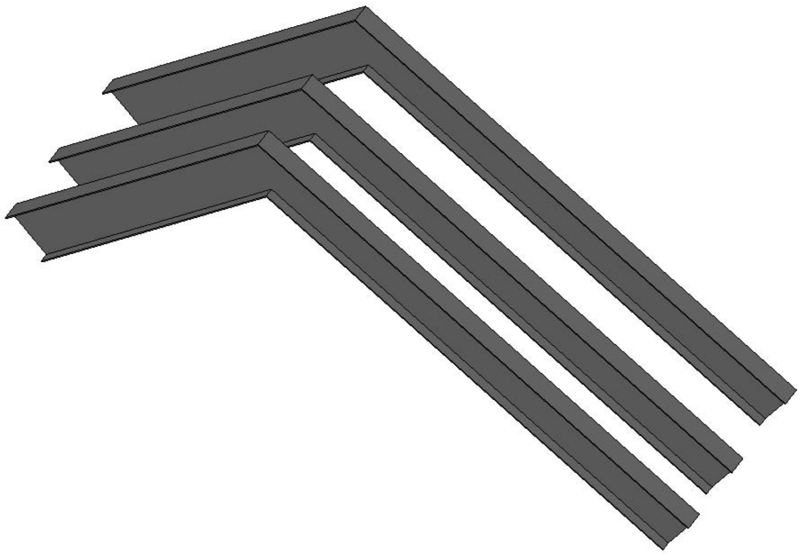
So do not get lulled asleep by your initial successes. When your project begins, try to scope out the elements that might cause you problems and prepare to deal with them. Let your boss know where you can expect difficulties and how much it will cost in time and effort to devise methods to deal with them. Don't get caught in the 80-20 trap.

We just worked through the process that you use to create a family from scratch. You got off to a good start because you used the template for beams and braces, which saved you a lot of work. You created the proper constraints for the tapered girder geometry. Then you overcame a big problem getting the sloping bottom flange to show the correct thickness. In the next section you will create a single in-place family within an existing project.

Bent Steel Beams

Another item you are bound to need while modeling a building is a bent beam (see Figure 19.10), whether in concrete or steel. Earlier, in the chapter on framing, you saw that you can create many types of curved beams, but bent beams require a different approach. You will probably be happy to learn that this one is not as involved as the tapered girder family. You will make an in-place family for the bent beam. Basically you create a solid sweep and apply a beam profile to it. Then you add a symbolic stick line symbol to finish out the family. Let us break down the process and step through its creation.

FIGURE 19.10
Bent beams in a 3D view



Here is how the procedure works. First you set up the file and start the in-place family:

1. Open a new default Revit Structure project. It should start with Level 2 open. Change the scale to **½″ = 1′-0″**.
2. Draw a single horizontal grid line above the south elevation mark (a named reference plane also works).
3. On the Design bar click the Modelling tab, and then choose Create.

4. Select Structural Framing from the Family Category list, and click OK.
5. Name the in-place family **Bent Beam 1** (you may have others to construct).
6. Click OK, which will fade the main model, and you can now begin constructing the bent beam.

Next you will create a solid sweep. You will need to use a plan view and an elevation view as you construct it. First you will use the elevation view to draw the path of the bent beam.

1. On the Family tab click Solid Form ➢ Solid Sweep, which will put you into Sketch mode.
2. Now double-click the south elevation in the Project Browser.
3. On the toolbar at the upper left side click Plane (Work Plane) and then choose Grid 1 as your active work plane (or select the named reference plane).
4. On the Sketch tab choose Sketch 2D Path.
5. Sketch the bent beam path something like the one shown in Figure 19.11.

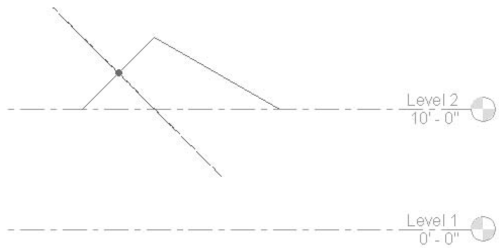


FIGURE 19.11
Sketching the 2D bent beam path in the elevation view

6. On the Sketch tab click Finish Path to complete the path.
7. Click Profile on the Sketch menu.
8. On the Options bar to the right of the Edit button, use the drop-down list to select W16X26 from the list of loaded profiles.
9. On the Options bar change the Angle value to **90** degrees in order to roll the profile into a web vertical position.
10. Click Finish Sweep, and the basic form is done.

Now you will apply a material to the bent beam.

1. Highlight the bent beam, and click the Element Properties button on the Options bar.
2. Click the Materials parameter field, which then displays the Materials dialog box.
3. Select Metal - Steel - ASTM A36.
4. On the right side of the dialog box click the Shading Swatch button, which displays the Color palette (see Figure 19.12).

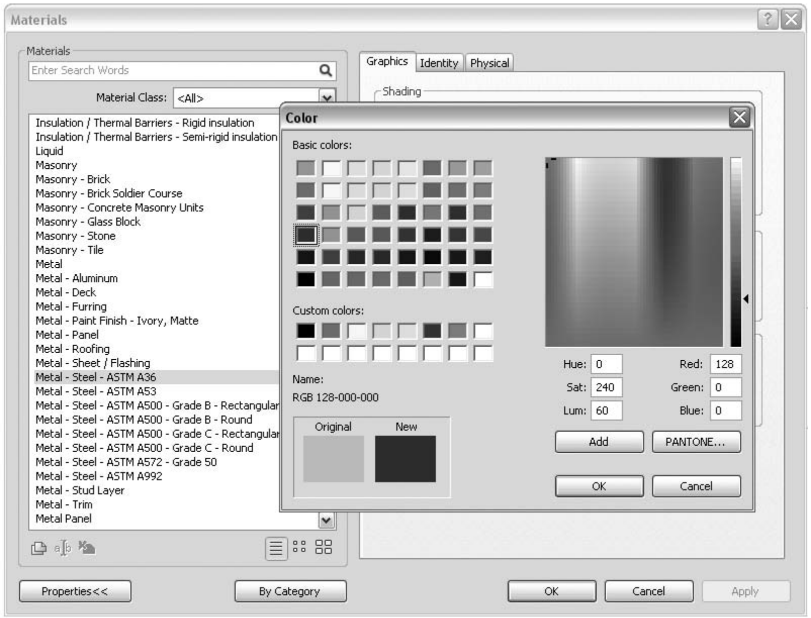


FIGURE 19.12
Changing the color for steel ASTM A36

5. Change the color to an RGB (red, green, blue) value of 128, 0, 0, a tone of dark red that makes the steel display more realistic.

6. Click OK until you exit all the dialog boxes.

You are still in the Family Editor, but now go to the 3D view and check out your results thus far. In the 3D view on the View Control bar click Model Graphic Style. Select Shading with Edges to get a good display of the bent beam. It is always a good idea to be checking the 3D view as you work to make sure it looks right.

Now you will add a symbolic stick symbol line in plan view. Go to Level 2, and type **ZE** to zoom extents. You will probably see something like Figure 19.13. Notice that part of the bent beam is above the cut plane of the view and not visible. You can ignore that or extend the view range up until it all shows.

FIGURE 19.13
The bent beam in plan view

To add the stick symbol to the in-place family, do the following:

1. On the Family tab choose Symbolic Lines.
2. In the Type Selector select Stick Symbols [Projection].
3. Draw the line along the length of the bent beam, right on the center.
4. Press the Esc key, and then highlight the stick symbol.
5. Close the locks at either end but not the middle one (see Figure 19.14).

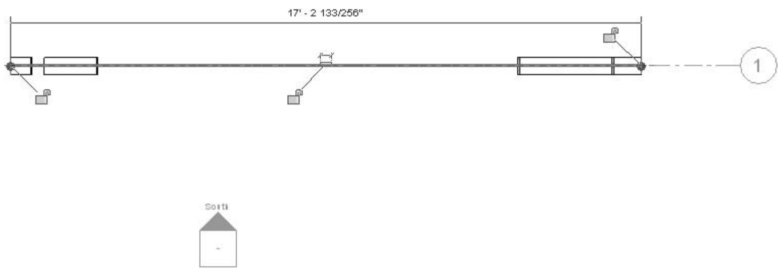


FIGURE 19.14
Locking the stick symbol to the model endpoints

Finally you will adjust the visibility of the family components so they show correctly in different detail modes.

1. With the stick symbol selected, click the Visibility button on the Options bar.
2. Uncheck Medium and Fine, as you do not want the stick symbol showing in those detail modes. Click OK to exit.
3. Select the bent beam sweep, and then click Visibility on the Options bar again.
4. Uncheck Coarse, as you do not want the sweep showing in your coarse views. Click OK to exit.
5. Click Finish Family.

You have now completed the process of creating an in-place bent beam family. The bent beam can be copied and adapted for other instances as required. The next section walks you through creation of a wood nailer on a steel beam family.

Wood Nailer on a Steel Beam

So far in this chapter you have created a parametrically driven tapered girder family and an in-place bent beam family. In this section you will alter an existing family by adding a wood nailer. What is that? If you work with wood products much, you will know (see Figure 19.15). Many commercial structures are built using multiple materials such as wood and light steel framing. In order to attach wood members such as floor framing and plywood sheathing to a steel beam, wood members are bolted to the top, and sometimes the bottom, flange of the beam. These are

called nailers. Revit Structure does not ship with a family to accommodate this assembly, so you are going to learn how to do it yourself.

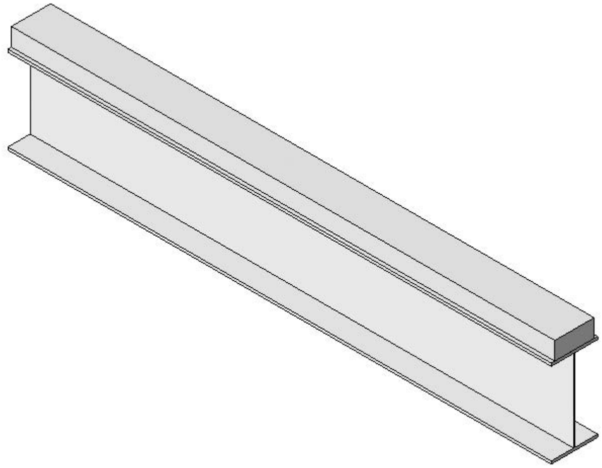


Figure 19.15
Wood nailer on the top of a steel girder

Now you could go about this by adding a wood member over a steel member independently of each other, but the process gets quite difficult to control that way. The best approach is to make your own family by adapting the basic wide flange steel family in order to make your own integrated wood nailer family.

Adapting the Shipped Revit Structure Families

Adapting the basic families shipped with Revit Structure can be a very useful approach to creating your own families, and it is a great way to learn more advanced approaches to your work. Just be sure that you do not save the adapted family with the same name, or you will end up overwriting the original family. Each family has an associated text file, which serves as a database of information. Each line in the text file is a record that describes a certain type, or distinct shape, within the family. You will need to copy that file to the new family name as well.

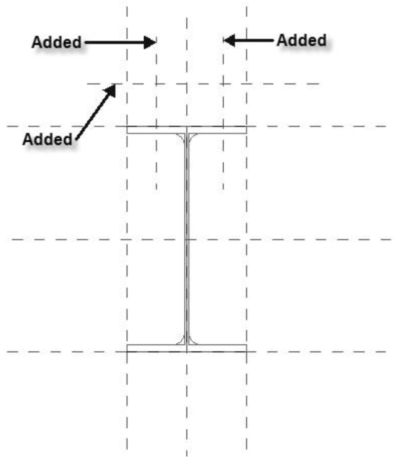
The procedure for creating the basic wood nailer family is as follows. First you need to set up the file:

1. Open a new default project.
2. On Level 2 draw a W16X26 beam.
3. Select the beam, and click Edit Family on the Options bar.
4. Click Yes to open the W-Wide Flange beam family for editing.
5. Immediately save the family using the Save As option. Name it **W-Wide Flange with Top Nailer**.

Using Windows Explorer, browse to your structural framing library in order to copy the text file associated with the family. Do you know where that is located? If you do not know, you can check under File Locations in the Options dialog box. The default location will be in your Documents and Settings folder. When you find the folder, do the following:

1. In the Steel Framing library copy `W-Wide Flange.txt` to `W-Wide Flange with Top Nailer.txt`.
2. Go back to the Revit Structure file, and choose the right elevation view.
3. Zoom in on the beam shape.
4. Draw three reference planes above the top flange in order to anchor the three sides of the nailer (see Figure 19.16).
5. Highlight the existing vertical center (Left/Right) reference plane, and click Pin on the toolbar.

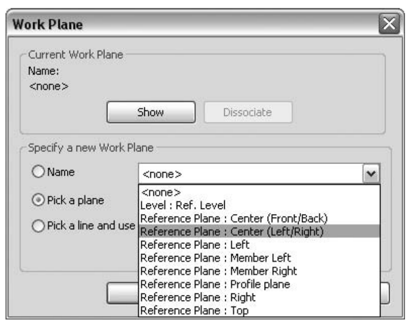
FIGURE 19.16
Draw three new reference planes above the top flange.



Now you will draw a solid extrusion that represents the nailer.

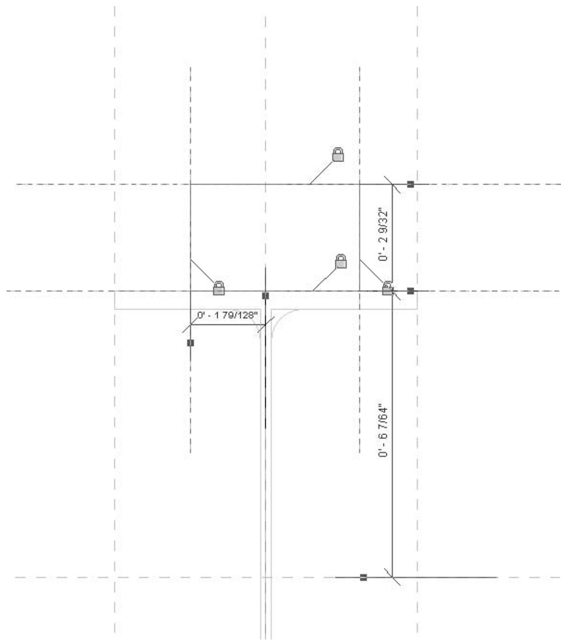
1. On the Family tab, click Solid Form ➢ Solid Extrusion.
2. For the reference plane select Reference Plane : Center (Left/Right), as shown in Figure 19.17.

FIGURE 19.17
Selecting the reference plane for the solid extrusion



3. Sketch a rectangle to represent the nailer, and lock the four sides to the reference planes (see Figure 19.18).
4. Click Finish Sketch.

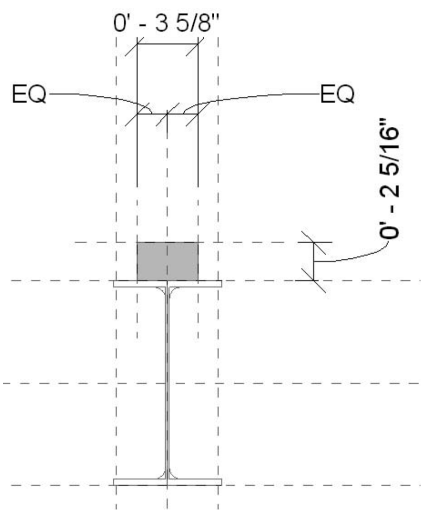
FIGURE 19.18
Lock the sketch lines to the reference planes.



Next you will add dimensions to the reference planes and then create labels that will control the nailer size.

1. Add dimensions to the added reference planes as shown in Figure 19.19.

FIGURE 19.19
Adding dimensions to the nailer reference planes



2. Select the Nailer Depth dimension.
3. Right-click and select Edit Label.
4. On the pull-down menu, select <Add Parameter>.
5. In the Parameter Properties dialog box, name the parameter **Nailer Depth.**
6. Group the parameter under Dimensions.
7. Click the Type radio button to make it a type parameter.
8. Click OK to complete the label.
9. Repeat steps 2 through 8 to create the Nailer Width dimension.

Now it is time to flex the nailer to make sure it works correctly.

1. On the Family tab click Family Types.
2. Under Dimensions change the Nailer Depth value to **2.5˝** and the Nailer Width value to **5.5˝.** Those values would be for a standard 3 × 6 profile of lumber.
3. Click Apply to demonstrate that the assembly can change to different shapes (see Figure 19.20). If you get a message that the shape is overconstrained, click Remove Constraints.
4. When you have finished testing, click OK to exit the dialog box.

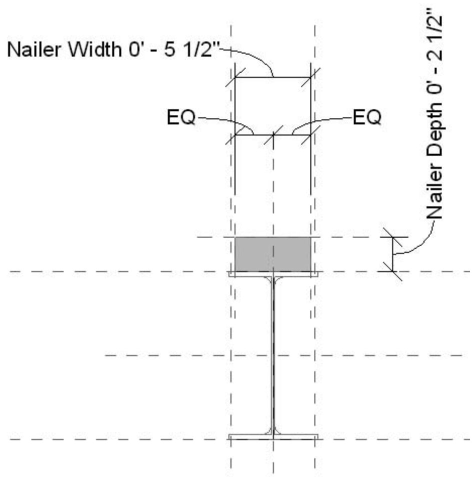


FIGURE 19.20
The dimensions have been changed to labels.

Now that the nailer profile is completed, the next step is to anchor the extents of it to the ends of the steel beam.

1. Go to the Ref. Level plan view.
2. On the Tools toolbar click Align.

3. Click the long reference plane at the beam end and then the one at the nailer end.
4. Close the lock, and then do the same for the other end.

Next you need to adjust the visibility of the nailer so that it does not show in coarse detail level.

1. Highlight the nailer.
2. On the Options bar click the Visibility button.
3. Uncheck the Coarse check box, and then click OK to exit.

Finally, you will assign a material to the nailer. Most likely there is no wood material loaded, so you will need to create it.

1. With the nailer highlighted, click the Element Properties button on the Options bar.
2. Choose the Material parameter, which will display the Materials dialog box.
3. Click the blue icon at the bottom left of the dialog box to create a new material by duplicating the default material.
4. Call the new material **Wood - Dimensional Lumber**.
5. Click the Shading tab, and set the RGB values to **223**, **192**, **134**.
6. Click OK until you exit all the dialog boxes.

You have now completed the family. Once again go to the right view, and then change the Model Graphics style to Shaded with Edges. Does it resemble Figure 19.21? Go to the 3D view, and flex the family with various nailer sizes to make sure everything is working correctly. Now you are ready to load it into your project whenever needed. One nice aspect is that the insertion will be from the top of the nailer, so setting its top in relation to your level can be done quite simply.

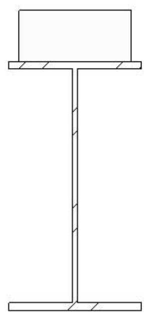


FIGURE 19.21
The finished nailer on a steel beam, forming an assembly

Wood Shear Walls

The next subject for you to explore is the creation of wood shear walls. Especially in seismic zones, the structural design and documentation of shear walls is essential to master. But what is the best practice in terms of modeling this type of element? It might be a retail store, or it might

be a student housing project. Your structural model will need interior and exterior shear wall types as well as bearing wall types. Wood projects can be quite a challenge to model because they have many, many pieces, as opposed to a steel or concrete building. So modeling them takes a lot of planning and good modeling techniques.

Of particular importance to this type of wall family is to embed 2D detail components right into the family. You will take a generic wall type and create a 6″ wood shear wall with the required elements. The process you will use to accomplish this is to create a profile family first and embed the 2D components into it. Then you will add that profile to the wall family.

Using Detail Components in Wall Families

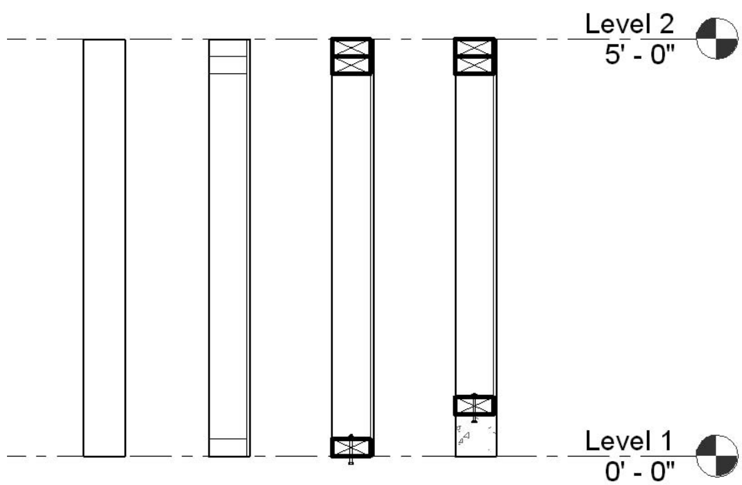
Using detail components in families is an important evolution in your practice. As you learn more about the family-creation process, you should look for ways to embed 2D components directly into the family.

The goal in doing so is to have the ability to cut a section through your model so that the result is as complete as possible. Having to add lots of 2D lines and detail components to your sections after they are placed in your project adds a lot of extra work, especially when the model starts flexing or you move the wall. The standalone 2D components you place in your sections tend not to follow the wall model and must be repositioned, oftentimes more than once. The wood shear wall is a good example of this. The shear wall will have plywood sheathing and top and bottom plates to which the studs are nailed. Why not have everything in one package?

To illustrate this idea look at Figure 19.22. There are four different wall types displayed in section view.

- On the left side is a generic 6″ wall that shows no detail of the wall. Lots of work to do there.
- Second from the left is a 6″ shear wall in which plywood sheathing has been added and cuts have been made to show the outline of the 3 × 6 nailers. That is a little better.
- Third from the left is a 6″ shear wall with plywood sheathing. It also has embedded detail components for the plates and the bolt at the bottom. Better still.
- On the far right side you see that a concrete curb has also been added as another option so you need not draw it separately .

FIGURE 19.22
Wood shear wall creation using detail components



So which wall type would you like to use on your project? Of course, you might ask why we are not modeling all the wall studs and plates. The answer is that we could but it would be very difficult to accurately place each stud. A worker building such a wall in the field would not find it all that useful to have such a model of framing plans. The important information is the overall design length, the sheathing location, and the hold-down anchor locations at the end of the wall and at doorways.

First you will create the profile for the plates similar to the way it was done in Chapter 10:

1. Browse to the Structural Library and choose Profiles ➢ Structural ➢ Wood ➢ Dimension Lumber-Profile.
2. On the Family tab click Family Types.
3. Click New, and name the new type **3x6**.
4. Set the b dimension to **5½″** and the d dimension to **2½″**; then click OK to exit.
5. On the Family tab click Detail Component.
6. Click Yes to load a detail items family.
7. Choose Detail Components ➢ Div 06-Wood and Plastic ➢ 061100-Wood Framing; then double-click Nominal Cut Lumber-Section.
8. Select a 3 × 6 component, and click OK to apply it and close the screen.
9. Rotate the component, and then move it directly over the profile lines.
10. On the Menu bar choose File ➢ Save As, and save the new profile with the name `Dimension Lumber-Profile with plate.rfa`.
11. Start a new default project, and go to Level 1. Save the project as **Wood Shear Wall Example**.
12. Go back to the new profile family, and load it into your new file.

Now you are ready to begin the process of creating the wood shear wall. Do the following:

1. In the new file on the Design bar choose Modelling ➢ Structural Wall.
2. In the Type Selector choose Basic Wall: Generic - 6″.
3. Click the Element Properties button on the Options bar to display the Element Properties dialog box for the wall.
4. Choose Edit ➢ New ➢ Duplicate.
5. Name the new wall **6″ Wood Shear Wall**, and then click OK.
6. Click Edit in the Structure field, which will display the Edit Assembly dialog box.
7. Click the Preview button on the bottom left to expand the view.
8. Change the view to Section: Modify Type Attributes.
9. Click the Material for Structure [1] area, and select Wood - Stud Layer. Then click OK to close.

10. Change the thickness to **5½″** (the actual size of a 3 × 6).
11. Click Insert to add the sheathing layer. Highlight the new layer, and click Down to move it out of the Core Boundary area.
12. Change the Material value to Wood - Sheathing - Plywood, and then give it a thickness of ½″.
13. Click Sweeps, and then in the Wall Sweeps dialog box click Add.
14. Set the following parameters for the sweep (see Figure 19.23):
    - Profile: Dimension Lumber - Profile Plate: 3 × 6
    - Material: Wood - Stud Layer
    - Distance: 1¼″
    - From: Base
    - Side: Exterior
    - Offset: -2¾″
15. Click OK to complete the addition of the sweep to the wall family.

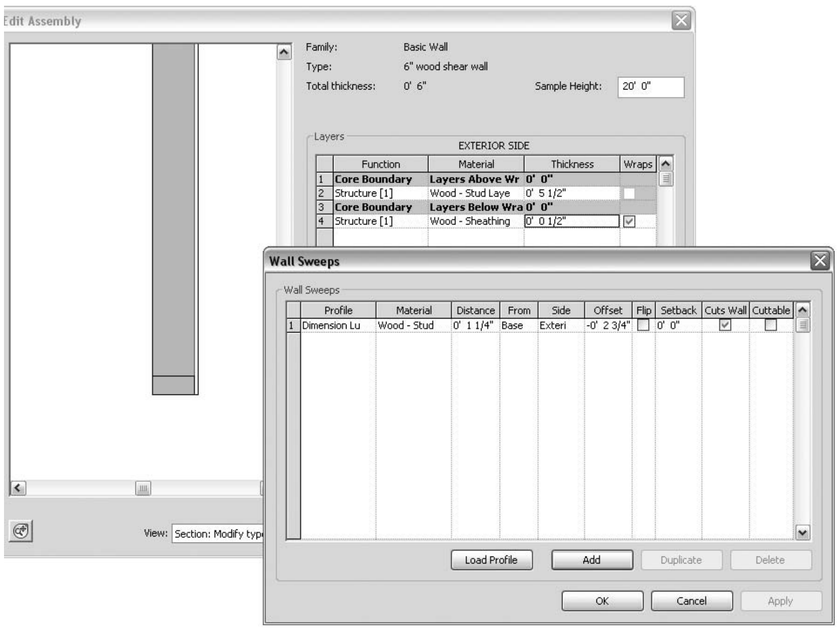
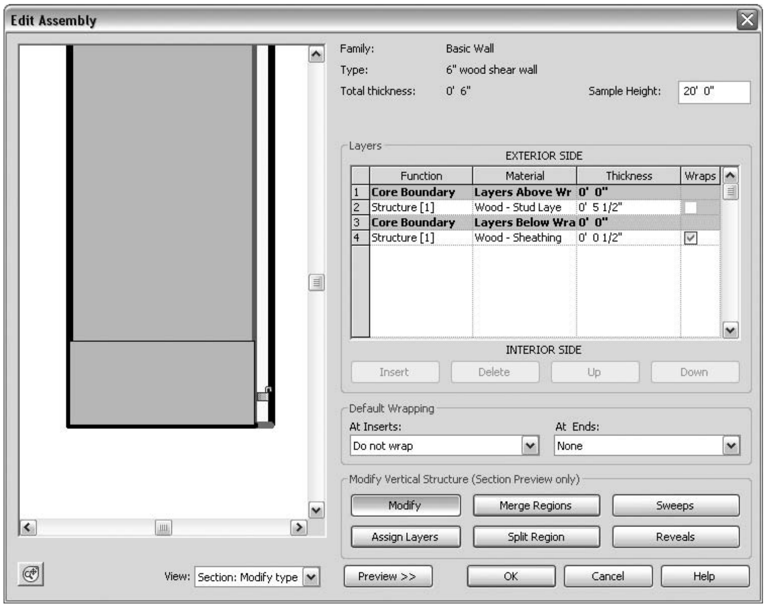


FIGURE 19.23 Creating the bottom plate for the shear wall

Next you will unlock the sheathing at the bottom of the wall so it can move independently in the vertical position when placed.

1. With the Sheathing layer still active click Modify, and then zoom up on the bottom of the wall in the preview.
2. Click the bottom line of the sheathing layer until you see the lock; then unlock it (see Figure 19.24).

FIGURE 19.24
Unlocking the sheathing layer



Next you will add two more profiles at the top of the wall to represent the top plates.

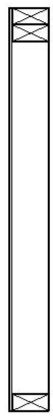
1. Click Sweeps and then Add.
2. Set the following parameters for the first top plate sweep:
   - Profile: Dimension Lumber - Profile Plate: 3 × 6
   - Material: Wood - Stud Layer
   - Distance: -1¼″
   - From: Top
   - Side: Exterior
   - Offset: -2¾″
3. Set the following parameters for the second top plate sweep:
   - Profile: Dimension Lumber - Profile Plate: 3 × 6
   - Material: Wood - Stud Layer

- Distance: -3¾″
- From: Top
- Side: Exterior
- Offset: -2¾″

4. Click OK to complete the addition of the two top plate sweeps to the wall family.
5. Click OK successively until you exit all dialog boxes.

Now draw a wall in plan view. Go to a section view, and change the detail mode to Medium. Your wall should look like that in Figure 19.25. Try pulling the sheathing layer up and down to make sure it is free to move. As a further example, you could create another profile family by adding a 2D detail component of a bolt in section to the file you created for the basic stud plate.

Figure 19.25
The completed shear wall with sheathing and top and bottom plates



There is a lot you can do there, and you will probably have to spend time getting used to this assembly. For now we will turn to creating an elevator pit family from scratch.

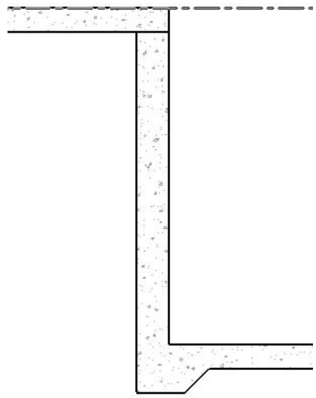
Elevator Pits

Almost every multistory building is going to have an elevator pit and most likely a sump pit. You will have to show the pit opening and then the pit walls dashed to represent the pit on your floor plan. The pit can appear in different sections you might take for detailing purposes as well. In the 3D modeling world you need to model the pit as one important structural component of the project. If you do not model the pit, you risk losing the integrity of the model as a BIM solution.

The first and most obvious way to approach the pit construction is to simply add foundation walls and a slab at the bottom of the pit. But this will take some effort in getting all the pieces situated correctly, and display can be messy. The pit slab and walls also generally intersect and form a bearing surface that is a little harder to construct (see Figure 19.26). The inner pit footing

line slopes up at an angle connecting to the pit slab. You will need to add a slab edge profile and then join all the geometry to get it looking right.

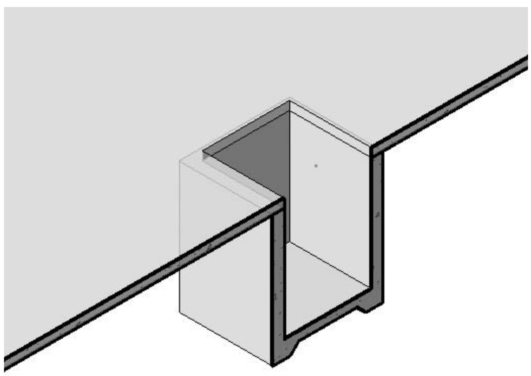
FIGURE 19.26
Elevator pit wall and slab edge condition



So instead of using that approach, what if you were to create an elevator pit family that could simply be loaded into your project and dropped onto the floor as a single unit? The 3D pit family would automatically attach to the underside of the floor slab and create the pit opening. The plan view would be exactly and cleanly displayed, and the whole thing could be pushed and pulled into place as one big assembly. For each project you could adapt the pit length, width, and depth dimensions to fit the particular installation in that building. Several types of pits could also be created on one floor.

Well, that elevator pit family is exactly what you are going to create in this section of the chapter (see Figure 19.27).

FIGURE 19.27
The elevator pit family in cutaway view



The procedure to create such a pit family can seem quite involved, so we will break it down into parts. You start a new family first and then add reference planes where the walls and pit slab will be located. You add dimensions to the reference planes that you later change to labels. Then you add solid extrusions for the walls and pit slab that are attached to the reference planes. Next you add void forms to make the floor opening and to sculpt the pit slab. Finally you add 2D symbolic lines and set the visibility for different detail levels.

To create an elevator family, follow these steps:

1. Open a new default project.
2. On the menu bar choose File ➢ New ➢ Family.
3. For the family template type select Generic Model Floor Based, and then click Open. You then will work on the floor that has already been created in the template. Using the floor-based template means that the pit will be hosted by a floor element.

Now you must establish reference planes to which you will attach the walls.

1. On the Family tab click Ref Plane.
2. Draw eight reference planes around the central axis to represent the pit wall lines (see Figure 19.28). Do not worry about exact placement.
3. Use Figure 19.28 as a guide to add dimensions to the reference planes. Lock only the four 8″ dimensions (you may want to use another value for the 8″ dimension).

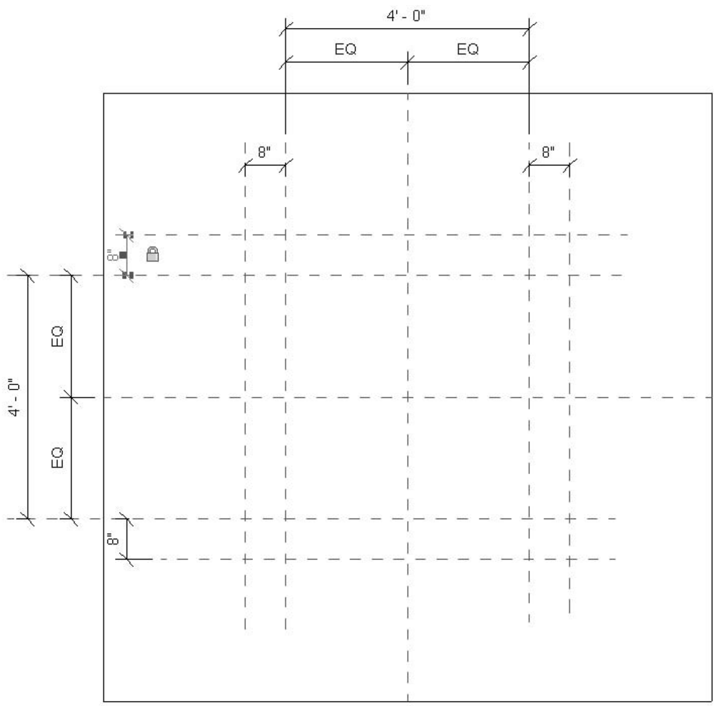


FIGURE 19.28
Create eight reference planes around the center point of the slab.

4. On the Family tab choose Solid Form ➢ Solid Extrusion.
5. On the Sketch tab choose Lines. On the Options bar select the Pick Lines option.
6. Now select each of the eight reference planes you just created, closing the lock after each one is selected (see Figure 19.29).
7. On the Tools toolbar choose the Trim command, and fillet the new lines to form boxes.

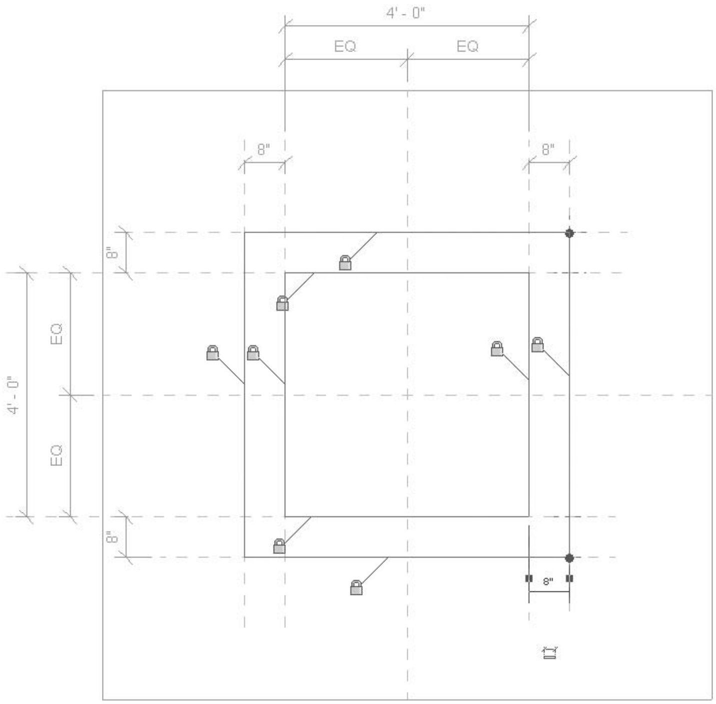


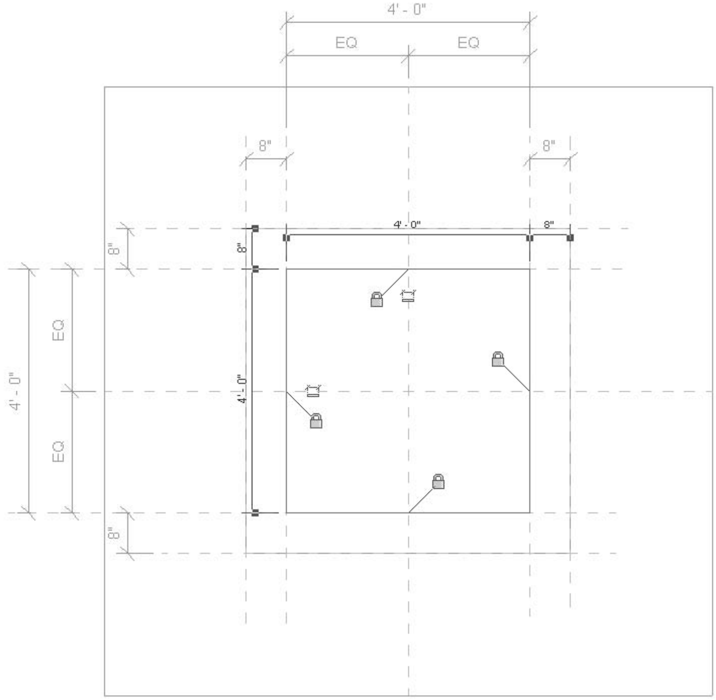
FIGURE 19.29 The wall lines are locked to the reference planes.

8. On the Family tab choose Extrusion Properties, and make the Extrusion End value equal to **-5´-0˝**. Click OK to close.
9. Click Finish Sketch to complete the walls.

Now you will create the pit slab:

1. On the Family tab click Solid Form ➢ Solid Extrusion.
2. Draw a rectangle around the inner walls, and lock them to the inner reference planes (see Figure 19.30).

FIGURE 19.30
The pit slab lines are now locked to the reference planes.



3. Choose Extrusion Properties to access the Element Properties dialog box.
4. Make the Extrusion End value **-5´-0˝** and the Extrusion Start value **-4´-0˝**.
5. Click OK, and then click Finish Sketch to complete the pit slab.

Next you will create the opening in the host floor slab. You should still be in the Ref. Level view.

1. On the Design bar click Opening, which places you into Sketch mode.
2. Click on Lines.
3. Click the Rectangle shape on the Options bar.
4. Create a rectangle around the inner walls of the pit and lock the lines.
5. Click Finish Sketch and there you have it.

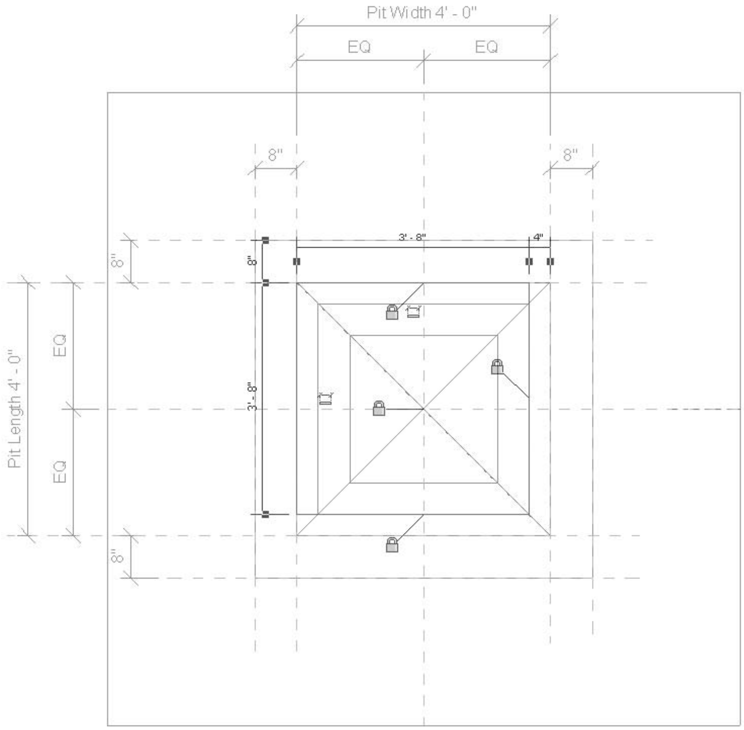


FIGURE 19.31
The slab opening is locked to the inner wall lines.

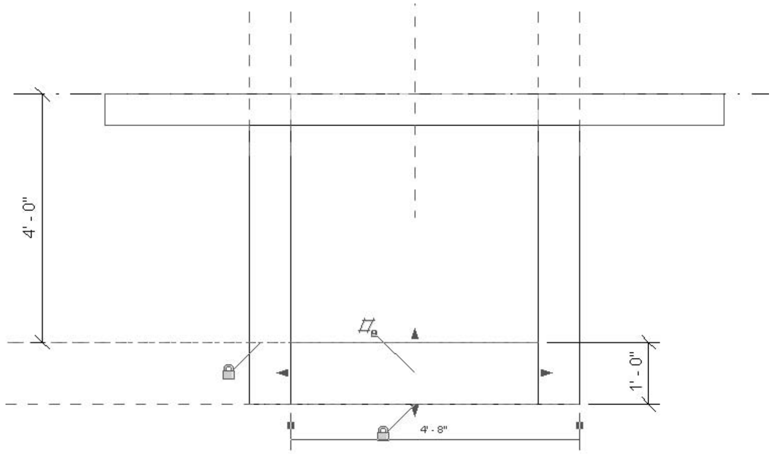
The next step is to add a reference plane and dimensions to control the depth of the pit.

1. Select Ref Plane on the Family tab and Pick Lines on the Options bar.
2. Click the top of the pit slab line. Stretch the new reference plane line out to the left.
3. Repeat step 2 for the bottom of the pit slab (see Figure 19.32).

Note that the bottom of the slab and the bottom of wall solid forms must both be locked into the bottom reference plane. If you are having problems selecting either one, press the Tab key until you can select properly.

4. Add one dimension from the floor line to the top of the pit slab and one for the thickness of the pit slab.
5. Adjust the top of pit slab depth to **4´-0˝**.
6. On the Tools toolbar click Join Geometry, and join the pit slab and walls.

FIGURE 19.32
The pit slab lines are locked to the reference planes.



The next step adds a solid void blend to carve out the bottom of the slab so its edges look like Figure 19.33. (The point is to make a bearing surface equal to 1´-0˝.)

1. Go back to the Ref. Level plan view.
2. On the Family tab choose Void Form ➢ Void Blend.
3. On the Sketch bar click Lines (if it is not active).
4. Select Rectangle, and make the Offset dimension equal to **-4˝**. Click the upper left and lower right of the pit opening to create the bottom of the blend, as shown in Figure 19.33.
5. On the Sketch bar click Edit Top.
6. Select Rectangle, and set the offset dimensions to **-10˝**. Click the upper left and lower right of the pit opening to create the top of the blend.
7. Choose Blend Properties, and set the First End value at **-5´-0˝** and the Second End value at **-4´-0˝**. Click OK to exit the dialog box.
8. Click Finish Sketch to complete the void blend.
9. Go to the front elevation again. Highlight the void blend, and align and lock the bottom to the bottom of pit slab (see Figure 19.34).
10. Add a reference plane for the top of the void, and lock the top of the blend to it. Make a **6˝** dimension, and close the dimension lock.

FIGURE 19.33
A solid void is added to sculpt the bottom of the pit slab.

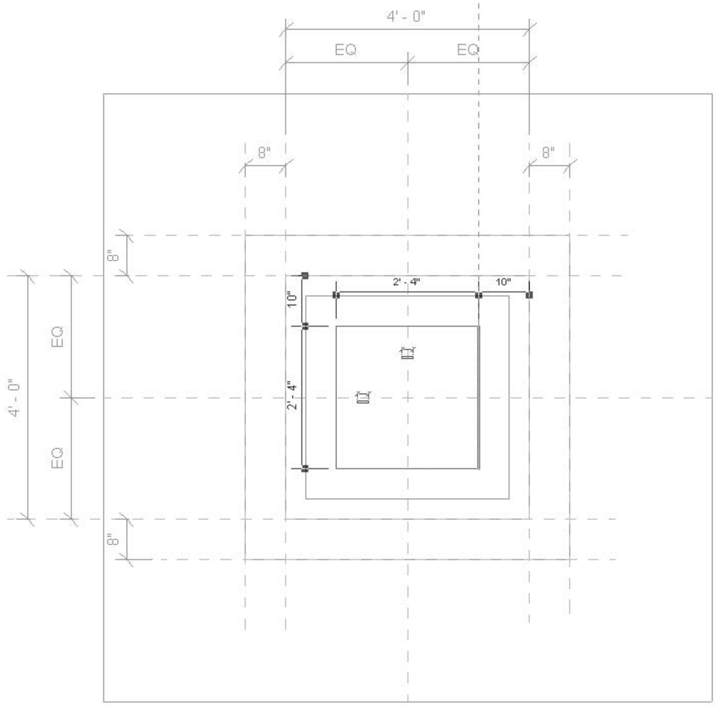
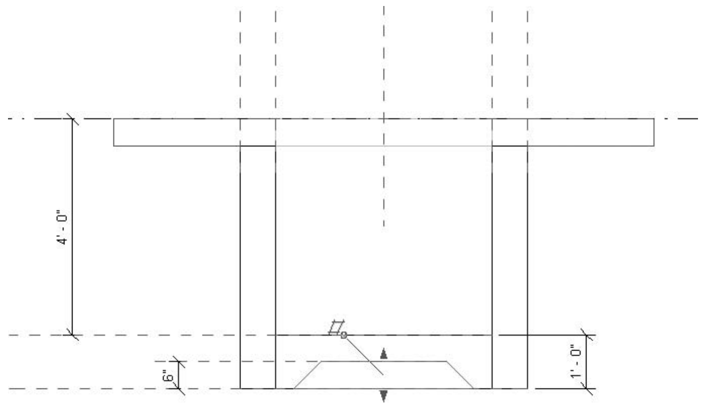


FIGURE 19.34
Locking the void to the reference plane



You have now completed the basic geometry of your elevator pit family. The final step is to make it parametric by changing the dimensions you have created into labels. The labels can contain different values that will represent the various pit types that you will create.

1. In the front elevation select the Pit Depth dimension, right-click it, and select Edit Label.
2. Click Add Parameter.
3. In the Parameter Properties dialog box name the parameter **Pit Depth**.

4. Group the parameter under Dimensions.
5. Click the Type radio button to make it a type parameter.
6. Click OK to complete the label (see Figure 19.35).

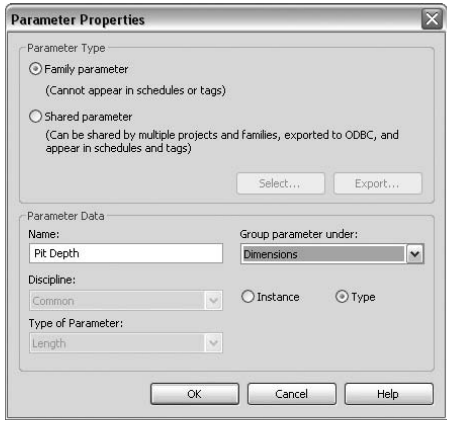


FIGURE 19.35 Defining the pit depth label in the Parameter Properties dialog box

Now you need to start flexing the pit model to test whether the lines you have created are locked into the reference planes. Remember that you need to constantly test your family by flexing the different dimensions you have established.

1. Click Family Types on the Family tab.
2. Change the Pit Depth parameter to **6′-0″**, and then click Apply.

Did the model stretch correctly to the new depth? If not, you need to exit and check the problem areas. Most likely something is overconstrained or improperly locked to a reference plane. This can get frustrating, but after awhile you will get the hang of it. Once you have the model flexing correctly, choose the Ref. Level plan view again.

1. Repeat the process, and create new labels for the Pit Z dimension and the Pit Y dimension.
2. Test them in the same way as you did the Pit Depth parameter to make sure the dimensions are flexing correctly to different sizes.

The final step to complete the elevator pit family is to add symbolic lines for your coarse mode and to establish the visibility of the various parts of the family. If you use the family the way it is now, the plan view will not display appropriately (see Figure 19.36).

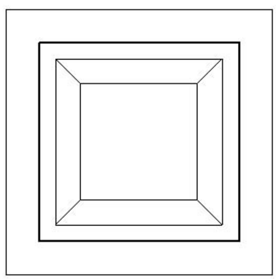


FIGURE 19.36 The final display must be configured using symbolic lines and visibility settings so it does not look like this.

So do the following:

1. Open the front view.
2. Select the walls, and then click the Visibility tab on the Options bar.
3. Uncheck the Coarse detail level, as shown in Figure 19.37. Click OK to exit.

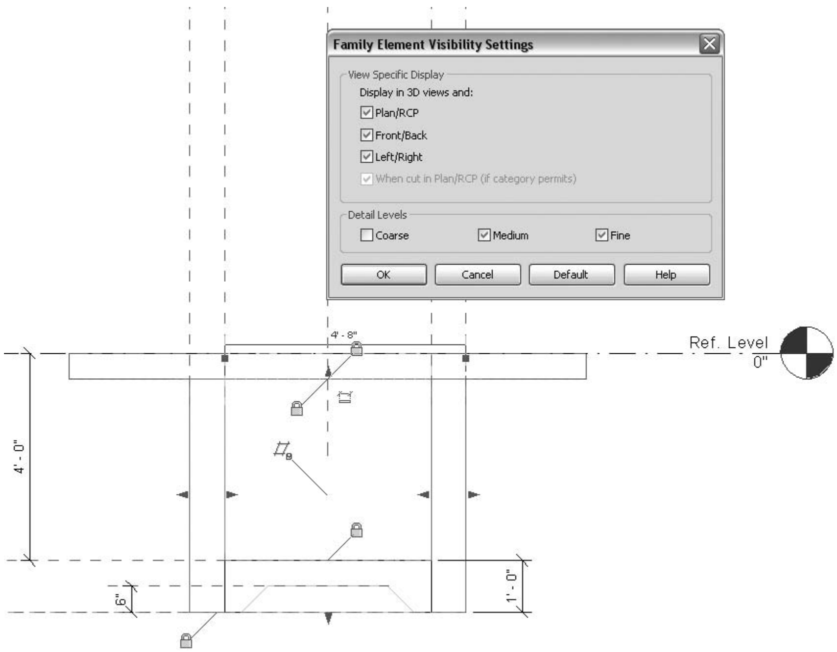


FIGURE 19.37 Use the Family Element Visibility Settings dialog box to set view-specific display characteristics for your family members.

4. Now do the same for the remaining solid and void forms.
5. Go back to the Ref. Level plan view.
6. On the Family tab click Symbolic Lines.
7. In the Type Selector field select Generic Models [Projection].
8. On the Options bar click the Visibility button.
9. Uncheck Medium and Fine so they do not show in those display modes.
10. Draw a rectangle at the edges of the pit, and lock the lines to those reference planes.
11. From the Type Selector select Hidden Lines [Projection].
12. Draw a rectangle at the outer edges of the pit walls, and lock the lines to those reference planes.

13. On the Family tab click Model Lines. You can use symbolic lines if you don't want to see an X in 3D views.
14. Draw an X with the Generic Models [Projection] type to represent the pit opening.

Now the pit in coarse mode will look like Figure 19.38.

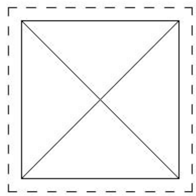


FIGURE 19.38
The pit plan view after the Visibility settings are completed, showing the symbolic/model lines in coarse mode

There you have it, a finished elevator pit family. It is time to take it for a test run. Start a new project, and on the first level add a 6″ concrete slab. Load the elevator pit family into the file. You will find the family under Generic Models. Drag it out, and drop it onto the floor (see Figure 19.39). Check out the plan view in coarse detail level to make it sure it is displaying correctly. Go to a 3D view, and notice how you can push and pull the pit around the slab. Take note as well how the walls automatically adjust to the floor depth and how the opening is automatically created in the floor slab.

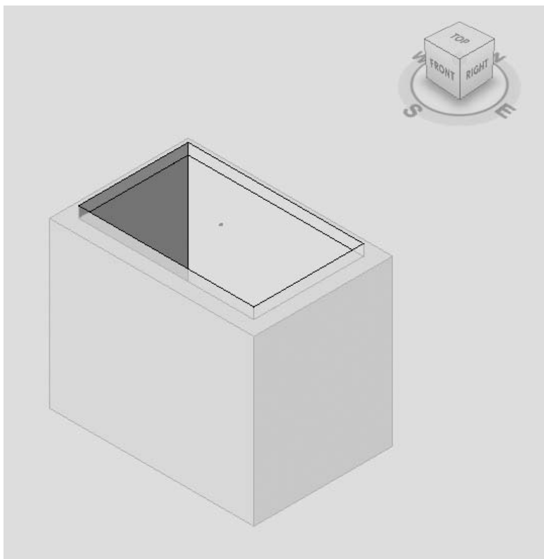


FIGURE 19.39
An instance of the elevator pit displayed in medium mode in a 3D view

You may now be getting a feel for the power of Revit Structure families and your ability to create your own families for the particular situations your project may demand. The next section will focus on another more advanced item, steel and wood trusses.

Steel and Wood Trusses

Trusses are one type of structural object that we did not cover in the chapter on structural framing. They tend to require more advanced thinking and planning. While there are many families available for insert into your model, you will come across many conditions that will require an in-place family solution. Trusses in Revit Structure are treated as assemblies that are created with numerous pieces. The truss families consist of basic truss forms (see Figure 19.40) and individual elements that can be configured with different structural framing members and whose spacing can be adjusted. The basic forms have a top chord and a bottom chord, as well as vertical and diagonal members.

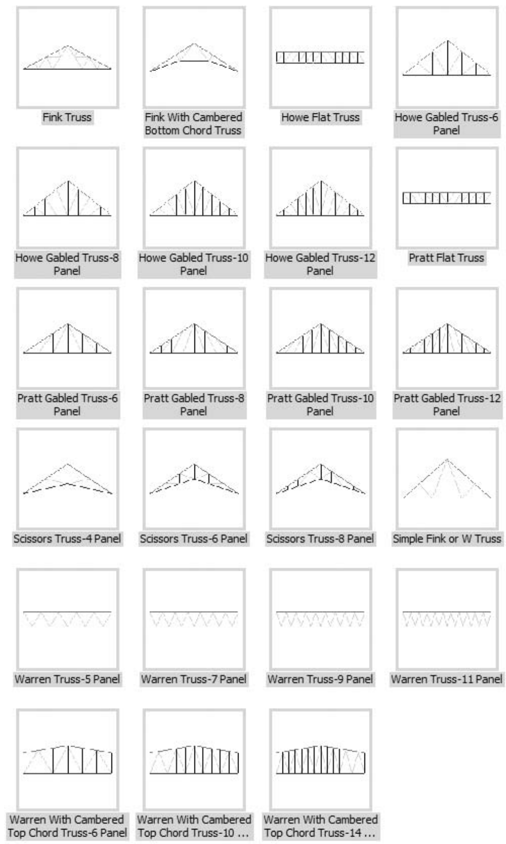
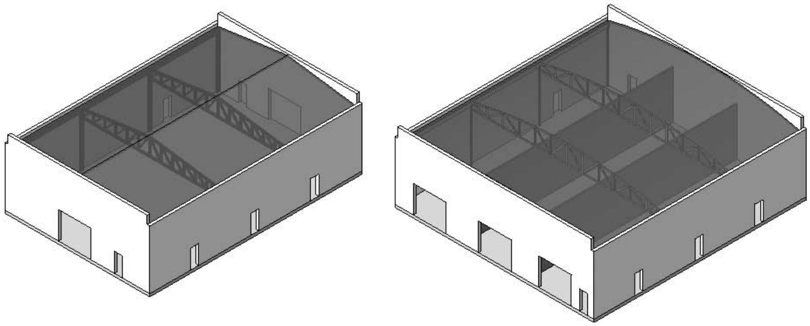


FIGURE 19.40
Various types of truss families available in the default libraries

In this section you will look at two warehouse buildings that require the addition of truss framing (see Figure 19.41). On one you will see how to add and create a truss whose top and bottom chords are WT shapes and whose web members are double angles as a free span over the entire warehouse floor area. This is a very common truss configuration. On the other warehouse you will create a wooden truss whose vertical members must be specifically located over intermediate bearing walls. That will require editing of the truss form itself, and it is the more difficult of the two examples.

FIGURE 19.41
Two examples of warehouses requiring truss framing

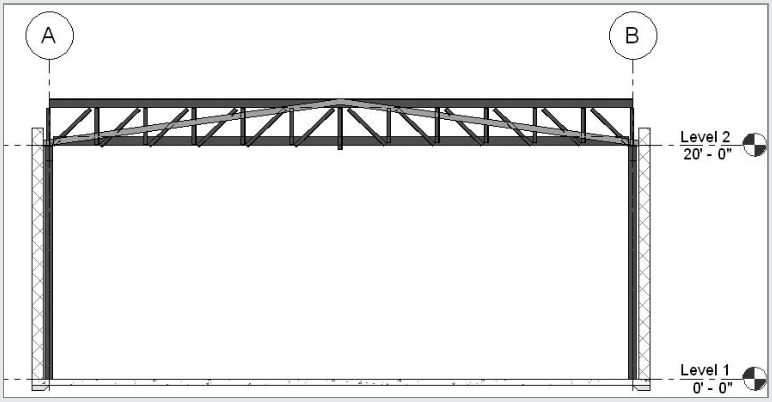


EXERCISE: ADDING A STEEL TRUSS TO YOUR PROJECT

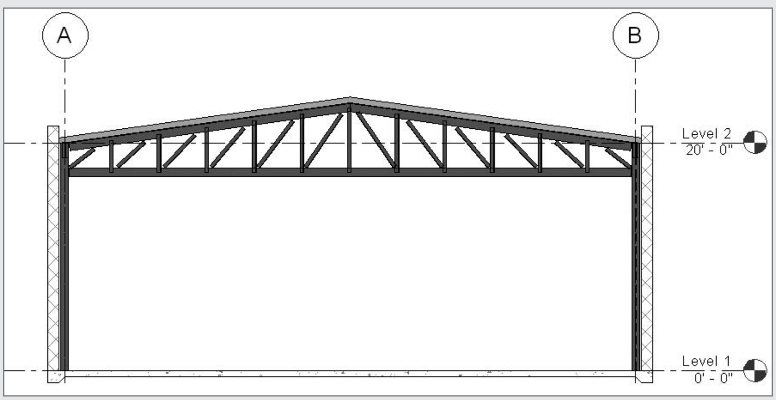
In the following exercise you will add a free-spanning steel truss to your project and then configure it for a particular condition.

1. Open Dataset_1901_begin.rvt (from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`)
2. In the Project Browser double-click Level 2.
3. Zoom closer to grid 2.
4. On the Design bar choose Modelling ➢ Truss.
5. In the Type Selector choose Howe Flat Truss : Standard if it is not set already.
6. On the Options bar click the Element Properties button.
7. Choose Edit ➢ New ➢ Duplicate.
8. Name the new type **WT+DBL Angle**, and then click OK.
9. For Top Chord Structural Framing Type select WT-Structural Tee:WT9X25.
10. For Vertical and Diagonal Webs Structural Framing Types choose LL-Double Angle:2L4X4X1/2.
11. For Bottom Chord Structural Framing Type choose WT-Structural Tee:WT9X25 and change the Angle parameter to **180**.
12. Click OK until you exit the dialog boxes.
13. Draw the truss from grid A2 to grid B2.

14. Now go to section 1, which should look like the following illustration.

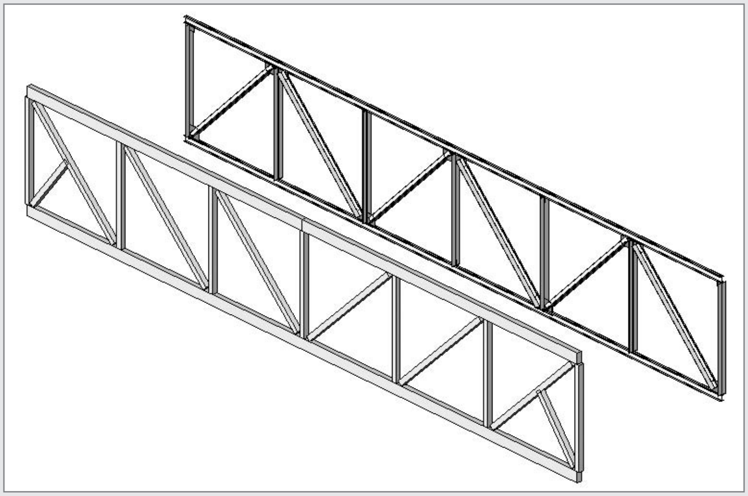


15. Highlight the truss form.
16. On the Options bar click the Element Properties button.
17. Change the Start Level and End Level offsets to **-2´-7½˝**.
18. OK to exit the dialog box.
19. Select the truss form, and then on the Options bar click the Attach/Detach Top Chord button. Select the sloping roof element. The truss form should realign with the roof.
20. Select one double angle, and then click the Element Properties button on the Options bar.
21. Click Edit/New, and then change the Space parameter to the width of the WT, 91/256˝, so the double angle fits over the WT web.
22. Click OK until you exit the dialog boxes.
23. Finally, highlight all the vertical double-angle web members.
24. Open the Element Properties dialog box, and change their Start and End Extension values to **4½˝** so they overlap the WTs.
25. Exit the dialog box, and you have completed the basic truss, which should look like the following illustration.



TINKERING WITH THE TRUSS MODEL

You can keep tinkering with the individual extensions and truss form to make it even more complete, as in the following illustration. Each member can be individually adjusted. For instance, each diagonal web member could be extended onto the WT web. Then you could cut the end so it is parallel to the chord member by using the Cut Geometry command. You can also add connection plates between the members. If you are working for a structural design firm and spending a lot of time on the plates, that might not make sense. You could document the connections with a typical detail and simply reference the elevation to those details. On the other hand, if you are working for a structural detailing firm or a contractor, you might want to develop each piece of the truss quite accurately.



But be careful not to overwork the object, and keep in mind the view scale at which you will display the model as well as your final documentation needs. Do not make the model more detailed than necessary. If you do, you will just be burning your project fee with little return to show.

Working with the Truss Template to Create a Wood Truss

When the built-in libraries cannot work for the condition required in your project, you can build your own truss family. On many projects you must specifically locate the vertical web members of the truss over intermediate supports, so you have to alter the basic spacing of the truss form. Using the truss family template makes the process much easier. The following explanation of this procedure will give you some first-hand experience in making your own truss.

In this second warehouse example the vertical web members of the wood truss need to located at third points of the truss span because the truss is supported by wood shear walls below. The structural design calls for placing a vertical web member directly over each one. The overall span of the truss is 70´-0˝, with the supporting shear walls at 23´-4˝ center to center. Develop the truss in the following way:

1. Open Dataset_1902.
2. On the menu bar choose File ➢ New ➢ Family.

3. For the family template type select Structural Trusses.
4. On the Truss Family tab click Family Types, and then in the Family Types dialog box change the Truss Length parameter to **70′-0″** and the Truss Height parameter to **4′-0″**. Click OK to close.
5. On the Truss Family tab select Truss Top Chord.
6. Sketch a line from the left reference plane to the right reference plane along the top reference plane. Lock the sketch line to the top reference plane.
7. On the Truss Family tab select Truss Bottom Chord.
8. Sketch a line from the left reference plane to the right reference plane along the bottom reference plane. Lock the sketch line to the bottom chord reference plane only.

Now that the top and bottom chords are completed, the web will be constructed starting with the critical vertical web members that go over the walls.

1. Add a vertical reference plane between the left and center reference planes.
2. Add a dimension string between the three reference planes and make them EQ (see Figure 19.42).

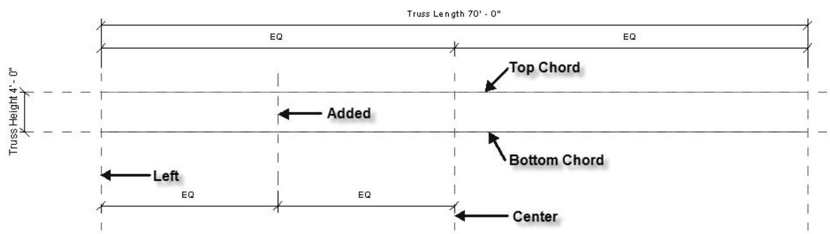


FIGURE 19.42
Adding the intermediate reference planes

3. Repeat step 2 for the right side of the truss center line.
4. On the Truss Family tab click Truss Web, and then sketch and lock vertical lines at the added reference planes between the top and bottom chords.
5. Now add five vertical truss web members between the two added reference planes and two on each side (do not worry about exact placement). Do not add vertical members at the left and right reference planes.
6. Using Figure 19.43 as a guide, add dimension strings to the added web members.

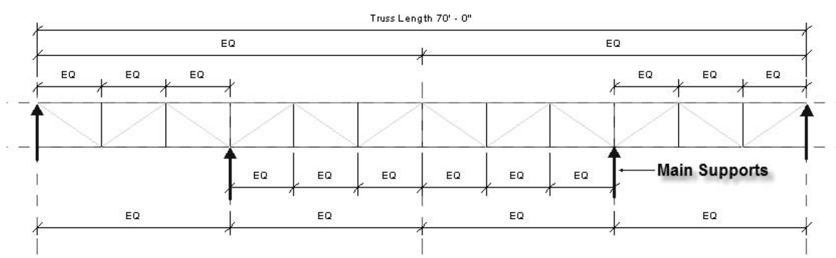


FIGURE 19.43
Adding the dimensions for the vertical web members in relation to the main supports

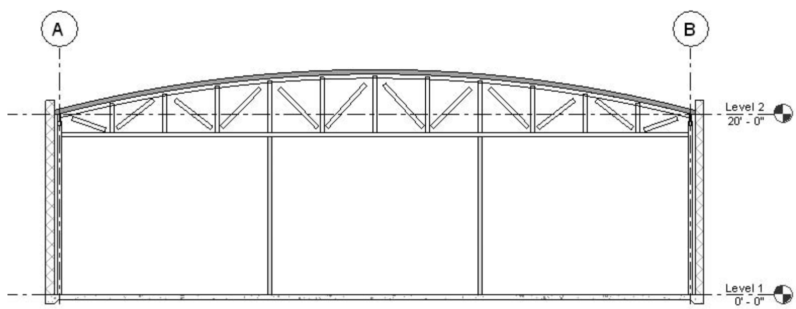
7. Again using Figure 19.43 as a guide, add diagonal truss web members between the vertical truss web members.
8. On the menu bar choose File ➢ Load From Library ➢ Load Family ➢ Structural ➢ Framing ➢ Wood ➢ Dimensional Lumber.
9. Click Open, and then select a 3 × 6 (which you will use for all the members of the truss), and click OK to load it into your project.
10. On the Truss Family tab choose Family Types, and then set the Structural Framing type for the Top, Bottom, Vertical, and Diagonal members to Dimensional Lumber : 3 × 6. Click OK to exit the dialog box.

The truss creation is completed. Now you will save and load the new truss family into your project and do the final configuration after adding it to the model:

1. Save your new truss family to your project directory with the name **Wood Truss Family**.
2. On the Truss Family tab click Load Into Projects.
3. Back in the building model, use the Project Browser to go to Level 2 if you are not there already.
4. In the Project Browser find the truss under Families ➢ Structural Trusses ➢ Wood Truss Family.
5. Draw the truss in plan view from grid A2 to grid B2.
6. Go to section 1.
7. Select the truss form, and click the Element Properties button on the Options bar.
8. Change the Start and End Level Offset value to **-2′-0″** and then click OK to exit the dialog box.
9. Highlight the truss form again.
10. On the Options bar click the Attach/Detach Top Chord button.
11. Select the barrel roof element, and the truss top chord will reconfigure to its radius.

Now the truss is completed and placed into your project. It should look like the one in Figure 19.44. You can further refine the vertical and diagonal member ends as discussed earlier, depending on your requirements. Note that the truss's vertical web members are accurately placed in relation to the walls below it.

FIGURE 19.44
The final placed wood truss girder



So that is the basic story on how to create, place, and configure structural trusses in a project. In the first case you inserted a truss form from the library and adjusted its spacing and members. Then you created your own truss from scratch using the truss template for a very specific web placement.

That completes this chapter on advanced structural families. There is incredible modeling power awaiting you when you get to this skill level. So always keep working on your technique, and try to take it to the next level. You will find that you can save lots of time and effort by taking a more sophisticated approach to your projects, using such families as we have presented here. You surely have some good ideas that will apply to your own work that are not included here.

The Bottom Line

Create a parametrically driven tapered steel girder family. In this first section you learned how to develop a new family from scratch using the Beam template. You used a blended sweep to create the taper on the lower flange of the girder. You had to introduce a number of new reference planes to which you could then attach the sketch lines of the girder. Adding dimensions between reference planes and making them labels gave you the ability to flex the shape and made the family truly parametric.

Master It What does "flexing the model" mean, and why is it important?

Construct an in-place bent beam family. The bent beam family was created inside the project as an example of an in-place family. You learned how to use a solid sweep form to sketch a path and a steel beam profile along a reference plane in order to create the desired shape.

Master It What members compose the bent beam family?

Adapt the steel wide flange beam family by adding a nailer to its top. In this section you adapted the existing wide flange beam family by adding an extrusion onto its top. The extrusion represents the wood nailer and has the ability to flex into different shapes.

Master It What are the four main steps necessary to add an extrusion in the shape of a 3×6 to the wide flange beam family?

Create an elevator pit family that can be dropped into your project. You learned to make an elevator pit family that could be easily inserted into your project. The pit form consists of solid extrusions and void extrusions working together. The whole family is floor based, so it can exist only when associated with a floor object in your model.

Master It What do the void extrusions do in the elevator pit family?

Produce wood and steel truss families using the truss template. In the final section you first worked through an exercise in which you inserted a truss from the structural truss library and then reconfigured it to create a steel truss with WT top and bottom chords and double-angle web members.

Then you used the truss template to create a wood truss from scratch so that its vertical web members could be specifically located above supporting walls in a warehouse.

Master It After you inserted the wood truss into your building model, how did you make the top chord of the truss follow the barrel-shaped roof?

Appendices

In this section you will find:

- **Appendix A: The Bottom Line**
- **Appendix B: Modeling Project Types**

Appendix A

The Bottom Line

Each of The Bottom Line sections in the chapters suggest exercises to deepen skills and understanding. Sometimes there is only one possible solution, but often you are encouraged to use your skills and creativity to create something that builds on what you know and lets you explore one of many possible solutions.

Chapter 1: Inside Revit Structure

Use the graphical user interface. The Revit Structure GUI is an easy way to interact with your computer in order to efficiently create your project model and documents.

Master It There are several ways to launch a single command in Revit Structure. List the various ways in which the Beam command can be invoked. Which method is the quickest?

Solution You can invoke the Beam command by selecting the beam icon on the Design bar (Basics or Modeling tab); by choosing Modeling ➢ Structural ➢ Beam from the menu bar; or by pressing BM on your keyboard.

One additional method that is often overlooked is the Create Similar command. If other beams already exist in the model, selecting a beam and executing this command will launch the Beam command.

You can find the Create Similar command on the Edit toolbar, on the context menu, on the menu bar (Edit ➢ Create Similar), and by pressing CS on your keyboard.

The quickest method to invoke the Beam command is by typing the default shortcut: BM.

Understand the types of elements in the modeling environment. In the modeling environment, there are basic types of model and annotation elements that you use in the construction of the virtual model and construction documents that you derive from the model.

Master It Modeled elements have a defined hierarchy that consists of categories, families, types, and instances. Use a structural column and give examples of each of these four element properties.

Solution The category is Structural Columns.

An example of a column family is the W-Wide Flange-Column W-Wide Flange-Column family.

An example of a type of column for this family is a W14X99.

An example of an instance of this type of column is the placement of a one two-story W14X99 column at a grid intersection in a project. Each column placement is a singular instance with its unique properties.

Create and manage project views. Even though you are building a 3D model, most of the time you are working in 2D views such as plans and sections. Therefore, the view types become your working planes and must be sensibly arranged.

Master It List all the major project view types discussed in this chapter.

Solution The project view types discussed were plans, callouts, sections, elevations, drafting, legends, schedules, 3D, sheets, and plan regions.

Control the graphical display of elements in a project. Creating the model is only half the story. Then you must derive the 2D and 3D views you will need for your construction documents. Each of these views must be able to display the model, and those display controls are an essential subject to understand.

Master It In your project you want to change the look of your masonry units on plans and elevations to match your company standards. Explain how to change the cut pattern for Concrete Masonry units to a diagonal pattern and the surface pattern to an 8 × 8 block.

Solution On the menu bar, click Settings ➤ Materials to open the Materials dialog box. Click into the Cut Pattern tab and change and change its setting to Diagonal Down. Click into the Surface Pattern tab and change its setting to Block 8 × 8.

Chapter 2: Setting the Project Environment

Develop your own custom templates Knowing all of the items that you can store in a template file will help you avoid creating data that is the same over and over and force company standards.

Master It What extension does a template file have? What types of things can be stored in a template file? What cannot be stored in a template?

Solution A template file takes on the extension of .rte. Just about everything that is in a project can be stored in a template file. Some basic examples are materials, line weights, fill patterns, view templates, project settings, and text and dimension styles. More advanced examples include wall types, slab types, schedules, and sheet setup. Worksets cannot be placed in a template file.

Set project units and precision display Setting the units for your project and the precision of them is part of documenting your model for those who will be using it.

Master It What are the three types of units in Revit Structure?

Solution The three types of units in Revit Structure are Common, Structural, and Electrical.

Make adjustments to structure-specific settings Revit Structure has several areas of settings that are specific to how elements display for your documentation.

Master It Where do you go to assign the global display of symbolic symbols for different types of connections? What type of family template do you use to create a new symbol type?

Solution To set the global display of symbols for different connection types, choose Tools ➢ Structural Settings. When creating a new symbol type for these connections, use the `Generic Annotation.rft` template with the appropriate category set to Brace in Plan View Symbols or Connection Symbols.

Organize the Project Browser Learning how to organize your browser depending on the workflow and requirements of the project will allow you to work more efficiently and keeps the browser free and clean of unused views.

Master It For what two things can you create a browser view type in the Project Browser? When creating a new project parameter so that you can sort your sheets in the Project Browser, what category do you need to apply the new parameter to?

Solution You can only create a browser view type for views and sheets in the Project Browser. When creating a new project parameter for sheets, you need to assign the parameter to the category Drawing Sheets.

Transfer standards into your project Being able to use settings and content from past or other current projects allows you to avoid duplicating your efforts over and over again and ensures that standards are kept.

Master It What command is used to bring new line weight settings into a project that is using old line weight settings?

Solution Use the Transfer Project Standards command (File ➢ Transfer Project Standards) to transfer standards from another template or project file into your current file.

Chapter 3: Starting to Model Your Project

Import and link CAD data. More than half of your projects are going to start with bringing in an architect's CAD data. In this chapter you learned how to bring a DWG file into your Revit Structure model and manipulate it to conform to your company's standards.

Master It Once you add a DWG to the model, you will find that the DWG does not look the way you would like it to. Name two methods of controlling the visibility of the underlay.

Solution By going to the Visibility/Graphics Overrides dialog box for that view, you can select the Imported Categories tab, where you can access all of the layers and change their display. You can also halftone the entire underlay.

Link Revit Architecture. The power of Revit Structure shines when you can get a Revit model from the architect. You can link that model and perform a Copy/Monitor operation to add superior integration unseen in CAD applications.

Master It Although the actual import of the Revit Architecture model is quite simple, you can copy specific objects from the Revit link and keep a live connection telling you if anything changes when the model reloads. How does Copy/Monitor work?

Solution Once the link is inserted, click the Copy/Monitor button on the Tools toolbar. You can then select the actual Revit link. The Design bar will transform into the Copy/Monitor bar. You can select either Copy or Monitor, depending on the situation.

Create structural grids. Revit Structure allows you to create grids "stick by stick." This freedom is crucial to being able to easily model any building shape needed.

Master It In this chapter you learned how to create a grid. Once the gridlines are in place, you have to make further adjustments. Explain how to do so.

Solution Once the grids are in place, select one. You will see a number of options available once a single gridline is selected. You can turn the grid head on or off, add an elbow, and even create a new grid style to leave out the middle portion.

Create levels. One of the most compelling aspects of Revit Structure is its ability to contain the entire model in a single file. The ability to create levels and generate floor plan views that are linked to them is a huge part of this functionality.

Master It As mentioned earlier, levels and plan views are connected. How does Revit Structure determine which level belongs to which plan view? What do you do if you need a new plan view based on an existing level?

Solution If you change the name of a level, you will be prompted whether you want to rename the associated plan view. If you click Yes, the plan view will keep the identical name as the level. If you find yourself in a situation where you added levels but not plan views, choose View ➢ New ➢ Floor Plan. The resulting dialog box lists levels that do not have views associated with them.

Chapter 4: Structural Columns

Work with the basic structural column family template Understanding the differences between the various templates for a structural column will help you ensure that your structural columns behave properly when placed into your project.

Master It What setting in a structural column family gives it the characteristics that help Revit Structure determine connection and attachment properties and whether the columns display in the Graphical Column Schedule?

Solution The Structural Material setting in the Structural Column family specifies how other framing members behave when they frame to them and how attachments behave when columns are attached to other elements. You can also use this setting to display only columns of certain material types in a GCS.

Place structural columns in your project Knowing all the methods available for placing structural columns in your project and knowing when to use each one will help you place columns quickly and accurately into your project.

Master It What are three methods you can use to place structural columns in your project, and how can you quickly place them onto your upper levels?

Solution You can put columns in your project by placing each column one at a time, or you can use the grid intersection method, which will put a column at each grid intersection you select. If architectural columns are in the project, you can place them by using the architectural column method. You can select and copy columns to the clipboard and then choose Paste Aligned to Other Levels.

Attach structural columns to other structural components To help maintain the top and bottom of a column's location and relationship to other elements, they can use different cut styles and justifications while being attached to various elements.

Master It What are four types of elements that structural columns can be attached to?

Solution Structural columns can be attached to floors, roofs, foundations, structural framing members, reference planes, and reference levels.

Employ the methods of placing a sloped column You can use several methods to place sloped columns into your project even though Revit Structure does not offer current families and programming that fully work with sloped columns.

Master It What are some of the methods used to create sloped columns?

Solution You can use several methods to place sloped columns in a project, such as using walls and editing their profiles, using sloped structural framing, or using in-place and external structural column families.

Document your model with the Graphical Column Schedule The Graphical Column Schedule can be generated automatically by Revit Structure and modified to prevent columns from showing up that you do not want displayed.

Master It What are three methods used to remove structural columns from the Graphical Column Schedule?

Solution You can use several methods to remove structural columns from the GCS. You can choose to not schedule off-grid columns, schedule by their material type, use phasing, or use filters to remove them.

Chapter 5: Floor Slabs and Roof Decks

Create a slab-on-grade with dropped slab edges. Using the Slab command, you saw how to create slabs-on-grade and to apply dropped edges to them with the Slab Edge function. You also learned how to edit the profile file for slab edges in order to create new types.

Master It You have a new project and have to add a slab-on-grade and slab edge at the bottom level. How do you do it?

Solution Using the Slab tool on the Modeling menu enables you to create a new slab type that is appropriate to your project. You may be required to make a new slab type depending on the actual one you are using.

Work with floor decks. You learned different methods for creating roof diaphragms and for editing them during the design process. You learned to create composite decks and how to create their geometry through sketching. You also learned to attach columns and walls to the roof diaphragm.

Master It Your project requires the metal deck to stop at the edge girder and the concrete to extend out one foot beyond to meet the inside face of the metal stud framing on the exterior. How would you create that?

Solution Create the floor deck type you need and create its geometry. While in Sketch mode, highlight the lines you wish to extend. Then on the Options bar select the concrete layer, and then enter the length you wish to extend it.

Work with warped roof decks. You learned different methods for creating roof diaphragms that are not just planar. Methods include using the slab subelement tools as well as creating more exotic roof shapes with solid modeling tools.

Master It Your project has a main ridge line across the middle of the roof with two drains on two edges of the roof diaphragm ⅓ points of the edge distance. How would you create it?

Solution In plan view, create the basic roof as a flat diaphragm. Highlight the deck, then add the lines and points from the subelement menu that will be used for the ridge line and drains. Adjust the values of the points and lines accordingly for your roof.

Create openings and depressions in your floors and roofs. You learned how to create shaft and incidental openings, and how to add depressions to the slab.

Master It On a multistory building, you want to add shafts to the core areas for stairs and elevators. How would you do that?

Solution On the Modeling menu, click Openings and then Shaft Openings to create consistent openings that automatically penetrate all floors and roofs. Constrain the shaft opening to the top and bottom levels, then extend as necessary to go through the slabs and decks.

Chapter 6: Walls

Place walls in your model. When you are adding walls to your Revit Structure model, you can rely on the fact that you will get both the width and height of the wall depending on that wall's type, as well as the height constraint you have set. Once the walls are placed in the model, you can easily adjust them to change when the building changes.

Master It Walls are quite simple to place in the model, but they can also be inaccurate if they are not added to the model deliberately. What are three things to look out for when placing walls in the model?

Solution The first thing to be sure of is the type of wall. Although you can change this later, it is good to at least be within the ballpark as to the overall width and structure. The second is the height offset, or the level to which the base and the top of the wall are set. Again, this can be changed later, but it will be much easier to adjust these parameters as you are placing the wall. The third is the justification of the wall. This is probably the most crucial because if this is wrong, you may not notice until you are dimensioning the walls. This could be at the end of the project once you discover the overall dimensions are slightly incorrect.

Create new walls. There is a good amount of functionality included in the process of building a new wall type. Furthermore, walls are a system family, which can be used as a basis for any additional wall type you may wish to create.

Master It The 14 walls in the default Revit Structure template are not going to be sufficient. Revit Structure provides the capability to modify a preconfigured wall system to suit your condition. Describe the procedures for:

- Creating a compound wall type
- Creating a stacked wall type

Solution To create a new wall system, you first must find a wall close to the wall you wish to create. You can locate that wall either by using the Wall command or by finding that wall in the Project Browser. In the Element Properties dialog box, click Edit/New, then click Duplicate (if you found the wall using the Wall command). Otherwise, click Duplicate if you double-clicked the wall from the Project Browser. Be sure the preview is opened. If it is a compound wall, you can start adding layers and materials. If it is a stacked wall, you can start stacking the predefined wall systems atop one another.

Modify walls in-place. When a wall has been placed, there are some additional functionalities allowed in Revit Structure for the modification of the walls that have been placed.

Master It Walls must conform to various conditions vertically, such as odd openings and a stepped base profile. Also, if you have a pitched roof, the tops of the walls need to be extended to meet the roof. Explain how these procedures can be performed.

Solution Once the wall you want to edit is placed, move to a straight elevation of that wall. Select the wall and look at the Options bar. If you would like to edit the profile (such as adding a footing step), click Edit Profile. Then, you can sketch the stepped profile as you please. If you need an opening, you can use the same edit profile method and sketch the opening, or you can go to the Modelling tab of the Design bar, and click Opening ➢ Wall Opening. If you need to physically attach the wall to a pitched roof or a sloping floor, select the wall and check the Options bar. You will see Attach, then choices for Top or Bottom. If you are choosing to attach the wall to a roof, click Top, and then select the roof. If it is a floor, select Bottom, then select the floor.

Chapter 7: Structural Framing

Understand structural framing families and properties. Revit Structure modeling is a constraints-based system that allows the model to update as changes occur, keeping the overall relationships between elements the same.

Master It Describe the two basic modeling constraints for attaching beams and braces and why they are important.

Solution The grids are the main horizontal constraint and levels are the main vertical constraints in your project. They can be updated through the life of the project and the model elements will stay attached and move with them. Framing members will be attached to their levels and to grids if they are on them, in such a way that the model will be able to flex when design changes occur.

Add floor framing. When you add floor framing to your project, you probably start with a fuzzy idea of the size and initially use a placeholder. As the design progresses and comes into sharper detail, you will update the sizes and spacing in many cases. The model must have a maximum of flexibility to make the editing practical.

Master It You are in schematics and know the bay widths on your building will change considerably. You want your framing members in each bay to be about 10´ from center to center no matter how wide the bay becomes during the course of the design. What layout rule is the best to use in this case?

Solution The best layout rule to use in this case is the Maximum Spacing rule. In that way, if the bay expands so that the members exceed the 10´ center-to-center setting, then additional framing members will automatically be inserted into the bay.

Add roof framing. Roof framing must support roofs that slope from ridges to drains. That means all the support beams and girders must slope as well. During the design process the roof can change in shape and slope. Costly editing can eat away your at your design fee.

Master It Calculating the end elevation for each sloping beam would be too time consuming and a nightmare to edit. What process do you use to most efficiently place the roof support system?

Solution Use the Beam tool with the 3D Snapping enabled. You select all the columns that intersect the roof diaphragm and then attach them to its underside. Next, add girders by snapping from column top to column top until the entire column grid is connected. The third step is to use the Beam System tool with 3D Snapping enabled so that the beams will intersect the sloping girders at top dead center.

Create moment and braced frames. Moment and braced frames are an important element of many structural designs. Revit Structure has two methods of displaying the braces in a plan views: Parallel Line and Line with Angle. The symbols are placed automatically in plan view as you draw the braces in elevation views.

Master It Which display type is the most informative of the braced frame layouts and how do you set it to display correctly in the plan view?

Solution Line with Angle is the most informative brace symbol because it shows the location and direction of the diagonal braces in plan view. A solid line for brace up and a dashed line for brace down can alert the design team to conflicts with doors or windows adjacent to the diagonal brace.

Chapter 8: Foundations

Create strip footings. This chapter illustrated how to best add continuous footings to the model. Strip footings are placed underneath a wall in a continuous manner. The walls that bear on these footings will dictate the location and length.

Master It There are two ways to create additional strip footing families in Revit Structure. One is to click the Element Properties button on the Options bar as you are placing the footing. What is the other way of creating a bearing footing as a wall foundation?

Solution In the Project Browser, go to the Families category. From there expand the Structural Foundations tree. Under Structural Foundations, expand the Wall Foundation tree. Now, you can right-click on an existing footing and duplicate it. Double-click on the new footing, and you can then modify the Type Parameters options to reflect your new footing.

Create footing steps. A stepped footing occurs in Revit Structure when you edit the profile of a wall to step into a graded area of a site. Additional families need to be added to complete the sequence. In this chapter, you learned how to modify a wall, and what actions to take to create a stepped footing condition.

Master It A footing will step with the wall, if the profile of the wall has been modified, but the step will simply drop with no angled bulkhead.

A) How do you edit the profile of a wall to step it?

B) How can you add a stepped footing family?

Solution To step a wall, go to the elevation where the wall is displayed. You can then select the wall, and click Edit Profile on the Options bar. Then, you can sketch the wall's stepped profile. If there is a footing already placed on the wall, it will be stepped along with the new profile.

To add an angled stepped footing family, the content will have to be created (or downloaded from the book's companion web page at `www.sybex.com/go/masteringrevitstructure2009`). You can then insert it as an isolated footing in plan. In elevation, you can move it into position.

Create and add foundation slabs. Foundation slabs are built in Revit Structure very literally. Slabs consist of material "layers," which in turn provide you with an overall slab thickness. In this chapter, you learned how to find a predefined foundation slab and modify it to suit your needs.

Master It How can a slab be modified to include a vapor barrier?

Solution A vapor barrier can be added to a slab by finding the slab in the Family category in the Project Browser (Families ➢ Structural Foundations ➢ Foundation Slab). Duplicate (or change the existing), slab, then go to the Type Properties dialog box by double-clicking on the slab. In the slab properties, click the Edit button in the Structure row. In the layers area, you can insert the vapor retarder under the slab.

Create grade beams. You place a grade beam in Revit Structure in the same way as you would a steel beam. Grade beams can be of any size and thickness. In this chapter, you learned the procedure for adding a grade beam to your model.

Master It Grade beams are added to the model as part of a foundation, but grade beams are added as an actual beam. What is the process for this?

Solution On the Basics tab (or Modelling tab), click the Beam button. Find a rectangular concrete beam from the Type Selector. Go to the Element Properties dialog box and click Edit/New. Then, select Duplicate. You can call the beam Grade Beam, and change its size. Although you will still be inserting this element as a beam, it will read as Grade Beam in the Type Selector as well as in the Project Browser.

Create elevator pits. Elevator pits are not an actual function within Revit Structure, but a compilation of structural items included in the foundation. In this chapter, you learned how to add an elevator pit to your model by combining several items.

Master It Which elevator pits are "formed" when a collection of elements are brought together? And how?

Solution When an elevator pit is created, the upper shaft is formed by (usually) CMU walls. Underneath the CMU walls there lies a pit wall. The slab runs continuously through the pit. The actual void for the pit is formed by adding a shaft opening (on the Modelling tab, select Opening ➢ Shaft Opening). The shaft opening is normally constrained to the bottom of the pit to the roof (or higher).

The bearing slab is typically a large isolated footing, which can be placed beneath the shaft.

Chapter 9: Model Documentation

Add datum elements to your detail and section views. Datum elements are necessary for your model because they are the anchors for your objects. Grids, dimensions, spot dimensions, and reference planes are basic constraints for elements within the model, and give it the ability to flex as you are working through changes in the design of the structure.

Master It Datum elements form the basic constraints for your project. Using Notepad, write a paragraph explaining how Revit Structure constrains the various modeling elements.

Solution Grids are the basic horizontal constraints to which your structural elements will be attached. They are tied together and controlled with dimensions so that they flex when the dimensions change. Levels are the basic vertical constraints that you use to attach tops and bottoms of columns, or beams to a particular level.

Add annotation elements such as text, tags, and symbols. Once the model is moving forward in development, you need to efficiently add identifying tags, beam annotations, and text to your various views in order to document your design and prepare your sheets. Tagging elments is an essential task since it taps into the properties of the object. If the object changes type, the tag automatically updates. That then allows you to use the model as a physical database for building schedules of many kinds. Text and symbols also are used to further the documentation of your model.

Master It Open `Dataset_0901_Begin.rvt`, then go to the second floor plan. On the second floor, load and tag all steel members. Add a Beam System tag to at least one bay. Add a Span Direction tag to the floor. Go to the first floor and tag the columns. After placing the tags, highlight and use the grips to align them with one another for a better display. Add grid dimensions.

Solution Load the tags by choosing Imperial Library ➢ Annotations ➢ Structural. Hover over the girders and click to add the beam tag. Make sure the Leader box is unchecked. For the bay in-fills, use the two types of Beam System tags. See `Dataset_0901_End.rvt` for the completed task.

Add detailing elements such as detailing lines and filled regions. Not everything is modeled. It takes experience to find the correct level of modeling in your project. For instance, columns are modeled but base plates are not in a typical American design firm. But when taking sections and creating details, you have to add that information in 2D over the modeled objects. So you add detailing lines to show the column base plate and perhaps some earth hatching around it. These are detailing elements.

Master It Open `Dataset_0902_Begin.rvt`, then go to the callout of Section 6. Add detail lines to show piping 41´-0˝ to the left of the column going through the slab, turning 90 degrees and going through the slab. Use a hidden line style. Add earth hatching below the slab using a filled region. Add a repeating CMU component wall to the right of the column with its outside flush with the grade beam below.

Solution Use detail lines, filled regions, and repeating details to complete the task. See `Dataset_0902_End.rvt` for the completed task.

Create a typical details library. A critical task to accomplish if you want your project to be totally documented in Revit Structure is the management of typical detail libraries. Typical details can be imported from your 2D CAD library or created from scratch in Revit Structure. You import Revit Structure details individually as drafting views, which are then added to sheets. They can also be inserted as part of a whole sheet. In similar fashion, you can export individual drafting views or sheets of drafting views to use in another job or to add to your Revit Structure library of details.

Master It You have a new project to start and want to transfer your model and drafting views from an already completed project. How will you transfer the drafting views to the new project? What is the best way to transfer a section with model elements in it to another project as a typical drafting view?

Solution You will transfer the existing drafting views by using the Save to Library command and exporting the sheets, which will also export and keep organized the drafting views it contains. Then in the new project, you will use the Insert from File command to import the views and sheets for the section that contains model elements. The best practice is to use the Freeze Drawing command to transfer all mode elements to drafting view lines. Then you can export and reimport that drafting view easily.

Chapter 10: Modeling Rebar

Draft a 2D rebar. In Revit Structure you can simply draft reinforcement by configuring line styles to be similar to an AutoCAD environment.

Master It Creating line work in Revit Structure is similar to drafting in AutoCAD. Name the process involved in getting the correct line weights and adding the lines to the model.

Solution Choose Settings ➢ Line Styles, and add and configure the lines you need. You can even mimic your AutoCAD line styles. Once the line styles are set, click the Drafting tab on the Design bar. Click the Detail Lines button and start drafting away! You can also click the Detail Component button to add "fake" rebars that are perpendicular to the view.

Configure rebar settings. In Revit Structure you can place reinforcement as actual objects as opposed to simple drafting. To do this correctly, however, you need to extensively configure the rebar settings for both graphics as well as performance.

Master It Walls, footings, and slabs have cover settings that allow you to place reinforcement in a more organized and accurate approach. How is this done?

Solution Choose Settings ➢ Cover Settings. In the resulting dialog box, you can add and change the cover settings needed for the project. Hopefully your company's template will have these settings preconfigured. To add these settings to objects in your model, simply select the object (such as a wall, slab, or footing), and click the Element Properties button on the Options bar. In the Element Properties dialog box, you will see the Structural category. In this category you'll find the cover settings needed to control the not-to-exceed rebar cover.

Draft a 3D rebar. Although Revit Structure uses a modeling approach, it is often necessary to be able to sketch reinforcement first, and then add it to the 3D Shape Browser once it is completed.

Master It Placing 3D reinforcement can be done in two different ways. Describe both.

Solution To place preconfigured reinforcement, you can simply select the item you want to reinforce, such as a wall, slab, or footing. Then, click the associated icon on the Options bar that will allow you to place rebars perpendicular to the current work plane, or parallel to the current work plane. Once you choose one, Revit Structure will display the Shape Browser. You can then select a shape and add it to the model.

You can also click the Sketch button on the Options bar. This allows you to draft any rebar shape. Once you have finished sketching the reinforcement, Revit Structure will add it to the Shape Browser.

Add rebar shapes. By default in Revit Structure 2009, you have a multitude of reinforcement to choose from. These shapes are preloaded into the template file you are using. Revit Structure allows for the importing of additional shapes.

Master It You may be working in a model that was created before the Revit Structure 2009 version release. The model will not have any rebar shapes. How do you import the shapes?

Solution Choose File ➢ Load from Library ➢ Load Family. In the resulting dialog box, browse to Rebar Shapes. Click in the Rebar Shapes folder, and press Ctrl+A. This will select the entire contents of the folder. Click Open, and the reinforcement will be brought into the model.

Chapter 11: Schedules and Quantities

Create Schedules Revit Structure has a good strong link between schedules and data. Once a schedule is created, the information can be manipulated either in the model or the schedule. Each will influence the next.

Master it Building a schedule can start in one of two places. One of the places involves using the View tab of the Design Bar, and selecting the Schedule/Quantities button. What is the second way of starting a schedule?

Solution If you go to View ➤ New ➤ Schedule/Quantities, you can then find the item you wish to schedule. Go through the schedule dialog tab-by-tab to configure the schedule the way you wish. If this is a schedule your company will use repeatedly, save it to a template, or alert your BIM manager that you have a nice new schedule to be used company wide.

Create Material Takeoffs Material takeoffs are virtually the definition of BIM. Revit Structure can really become an advantage to project costing in a real-time sequence. Once the tools are developed, and the basic takeoffs are in place, you can really start to see the benefits of taking the time to learn this feature.

Master it Creating a material takeoff is different than creating a schedule; however, it is similar in many ways. What is the major difference between creating a material takeoff and creating a schedule?

Solution In a material takeoff, the materials are listed as separate items as Material: <material property>. This type of data is not present in a normal schedule. Other than the additional properties of Material: the two items (Schedules and Material Takeoffs) are comparatively identical.

Create Legend Schedules Legend schedules are a great way to sequence items in Revit. The nice part about this functionality is that you can simply draft items as well. This gives the user much more flexibility as they create legends for their model.

Master it A legend schedule is tied directly into the item being scheduled. What is the procedure for adding a legend schedule number to an item?

Solution Go to View ➤ New ➤ Schedule/Quantities, then select the item you wish to add to a key. To the right of the Category field, select Schedule Key and click OK. In the Fields dialog, add the Comments field and click OK. This brings you to the view layout. From here right-click and select New Row. The row will receive an automatic number. You can type whatever comment you wish. Close the schedule view, and select an item in the model that corresponds with the key schedule. Go to the element properties of the item. In the Instance Parameters, under Identity Data, select the Style (if this is a foundation, for example, it will be listed as Structural Foundation Style) and change it to the key value you wish. The comment field will be filled in as well, only it will be unelectable in this dialog.

Send Schedules to Microsoft Excel Keeping track of the quantities may not be done by the designer. Many times it is an estimator who is not involved with the modeling process at all. This functionality allows the designer to output accurate data to an estimator in a format they are used to.

Master it Once a schedule or material takeoff is created in Revit Structure, how is it exported to Excel?

Solution First create a schedule or material takeoff in Revit Structure. Making sure the schedule or material takeoff is the current view, (or at least the view needs to be open) go to File ➤ Export ➤ Schedule. Browse for a place to save the .txt file. At the next dialog, accept the defaults as they appear. Next, open Microsoft Excel, and open the .txt file (You will have to change the Files of Type at the bottom of the dialog). A Text Import wizard will appear. Select the defaults and pick the Next button for the three dialogs of the wizard. Once you are finished, Excel will display the data. At that point, you can adjust the cells so that you can clearly read the data. Now you can save the file as an Excel (.xls or .xlsx) format. Also remember to save this file in the job directory so the other team members will have access to the data as well.

Chapter 12: Sheets

Create a Titleblock to Display Project Information The basics of creating a titleblock include using line work, annotations, filled regions, labels, and images. Combining these basic elements to create parametric behavior will take you way beyond 2D drafting.

Master It What are three ways you can make your titleblocks parametric to autoadapt to changes that are made within your sheets?

Solution Three potential ways to build parametric behavior into your titleblocks are to add labels that display project information, add Yes/No parameters to control the display of Not for Construction, and add a revision schedule to track revisions made to sheets.

Create a Revision Schedule to Your Company Standards Revision schedules added to titleblocks allow you to keep track of revisions on sheets. You can design these revision schedules to accommodate just about any company standard and titleblock configuration.

Master It How do you rotate a revision schedule 90 degrees? Where do you go to set a revision schedule to display its information from the bottom up?

Solution To rotate a revision schedule in a titleblock, select one that is already placed on a sheet. With the revision schedule selected, set its rotation by choosing 90 degrees clockwise or 90 degrees counterclockwise from the Options bar.

Explore the Behavior of the Various View Types When Placed on a Sheet When views are placed on sheets, new parameters become available that display information that is specific to how and where the views are placed. Each view can have a different behavior; knowing this behavior allows you to take advantage of it.

Master It What are four parameters that become available in plan, elevation, detail, drafting, and 3D views when they are placed on a sheet? What types of views can be placed on a sheet more than once without being duplicated?

Solution Several parameters become available for plan, elevation, detail, drafting, and 3D views when they are placed on a sheet. They are Rotation On Sheet, Title On Sheet, Detail Number, Sheet Number, Sheet Name, Referencing Sheet, and Referencing Detail.

Most views can be placed on only one sheet to allow Revit Structure to keep track of their location within the model. Some views do not require this behavior, so they can be placed onto more than one sheet. They are schedules, Graphical Column Schedules, and legends.

Produce a Sheet Index to Keep Track of Your Issued Sheets Revit Structure lets you easily create a sheet index to keep track of sheets.

Master It How can you add a second element to a sheet index to manually account for issue names and sheets that are being issued?

Solution You can add additional parameters to the Drawing Sheets category, which can be added to the sheet index schedules. You can add characters for the values of these parameters to denote the sheets being issued. By using a drafting view you can create a schedule with line work and annotation, and you can place it alongside to work in conjunction with the sheet index.

Control the Behavior of Revisions in Your Project You can control the tracking of revisions made to the model to reflect several standards that may be required as well as react to unknown project schedule changes.

Master It How do you set a revision tag to display an alphabetic numbering standard, and how do you rearrange a revision to be put on hold and placed behind another?

Solution Companies using the alphabetic numbering standard can set this behavior on a per-revision instance by choosing Settings ➢ Revisions and setting the numbering method to Alphabetic in the Sheet Issues/Revisions dialog box. The same dialog offers the Move Up, Move Down, Merge Up, and Merge Down buttons for rearranging revisions.

Chapter 13: Worksharing

Determine when to enable worksharing. Looking past day one of your project to help determine the proper game plan for moving forward will make things go much smoother.

Master It What can determine when you should enable worksharing?

Solution If more than one person will be repeatedly working on a project, you should enable worksharing to avoid getting file read-only warnings. Large projects will need to be broken into worksets to increase performance. The ability to perform standards checking parallel to those modeling will increase productive.

Enable and set up the worksharing environment. When working in Revit Structure, you'll almost always be in a multiuser environment. Knowing how to enable worksets and use the central and local files is important for communicating between team members.

Master It How do you enable worksets and where should the central and local files be saved?

Solution Worksets can be enabled by choosing File ➢ Worksets or by using the icon on the Worksets toolbar. After worksets are enabled, create a central file on the network. Users should then create and work on local files that are saved onto their workstations' hard drive.

Request and grant permission of elements. Working in a multiuser environment where you are sharing a project with ownership rights will eventually lead to team members tripping over one another. Understanding how the ownership rules are set is important when working efficiently.

Master It What are the methods used so you can take ownership of an element(s)? What do you do if another team member has ownership of an element(s) that you need?

Solution Borrowing individual elements is preferred over checking out worksets. Elements are borrowed automatically as you edit objects, but it can be done manually by selecting the blue puzzle piece or by right-clicking on the object and selecting Make Element Editable. If another user has ownership of an element that you need, you can request permission by making an editing request.

Stay in sync with other team members. Creating the central file and communicating to it with a local file is how you effectively work in a workshared environment.

Master It What is the recommended way to create a local file? What commands are used to get information back and forth between the central and local files?

Solution The preferred method to create a local file is to use the copy/paste/rename features through Windows Explorer. Once working in the local file, you use the Save to Central and the Reload Latest commands to stay in sync with other users.

Properly maintain your project file. Keeping your central file as healthy as possible will decrease chances of file corruption and increase overall performance.

Master It When should you audit the central file? How should you go about upgrading your file to a new release?

Solution Performing an audit at least once a month to the central file will help keep it healthy and minimize file corruption. Prior to upgrading to a new release, create a backup file and do an audit on the central file. The audit of the central file should be done by using the Revit Structure version that it is currently saved in, not at the same time as an upgrade is taking place.

Chapter 14: Visualization

Determine what and when to model. Once you get going in Revit Structure, the ease of creating models is both a blessing and a curse. If you model too little, you don't achieve the desired result. If you model too much, then you will have so much more than you need, your renderings will take an excessive amount of time.

Master It Before modeling, develop a scope of what and when to model. Conduct team meetings with all project modelers so that everyone involved has rendering in mind as they do their work. Limit the complexity of your renderings by using appropriate detail levels.

Solution Create a project model document with instructions for what and what not to model. This will include primary beam members as well as secondary and bracing elements. Using the Choose View menu, choose New ➤ Default 3D View. Then name them your view specifically for rendering purposes. Using Use the Detail level control at the bottom of the new 3D view and set it to the Medium level for entire building views. For views that are connection specific, assign a Fine detail level so that complete member profiles are seen visible.

Assign materials to your model. Actually rendering in Revit Structure isn't hard—Having something render-worthy is the hard part. Materials really do make or break your renderings. You can make your model look real, or like a real model.

Master It As you develop your families, assign materials so that you can render on demand later. Using the Materials dialog box, create materials for steel and concrete for when they are viewed at a distance. Adjust materials for rendering even if you won't be rendering now. This will reduce the time needed to prepare for when you are asked to produce images. For real photographic needs, use materials that have few repeating patterns so that no matter the point of view you use the materials you use will still maintain a level of smoothness.

Solution Using the Choose Settings ➤ menu choose Materials. Locate Concrete — Cast-in-Place Concrete and then select it. Then using the Render Appearance tab, click the Replace button. Then in the Render Appearance Library dialog box, type Concrete as the search content. Then choose the single Concrete slide and click OK to accept. Click OK again, and then you'll see that all basic concrete has a cleaner rendered look. For steel, once again use the Material dialog box, but this time look for Metal – Steel – ASTM A992: this is the typical material for major steel elements. Then in the Render Appearance Library, select Paint Dark Red Matte. Once found, select it and click OK twice.

Define the quality and style of your renderings. When you begin to render your model, you can be overwhelmed with all the settings at your disposal. You can define where the Sun is, what time of day it is, what resolution to create, and how detailed your images should be.

Master It Take a look around in the real world. Get a sense of what structures look like when they are under construction. Things are often dark; you don't always have to light everything up. When you create renderings, save time and only create high high-quality at the very end. Use the Rendering system with Autodesk mental ray to define a Sun, adjust exposure, control shadows, and create renderings. Then save your rendering to any number of image types.

Solution Using a 3D view, click the Teapot icon at the bottom of the view to open the Rendering dialog box. Use Using the Setting drop-down list, and select Medium. Under Lighting, change the direction of the Sun to Sunlight from Top Left. Change the Background Style to be Sky: No Clouds. Now click the Render button at the top of the Rendering dialog box. Once the process is complete, click the Adjust Exposure button and click Reset to Default at the top. Then change the Shadows value to 3 and click OK to close. Finally, click Export and save your rendered image to a JPEG format.

Export your models for other uses. Exporting your model for outside use is a typical activity of the true professional. You don't use one kind of writing implement, so you should not use only one rendering application.

Master It Once you have a 3D view active, you can export it to a DWG or FBX file to use in an outside application. Use the FBX format if you have Revit Structure cameras you want to export as well. Use the DWG format with polymesh for direct import into 3ds Max Design. But if you have very large models, you might want to use ACIS solids as since that allows 3ds Max to control the meshing directly.

Solution Using the Choose File menu choose Export ➤ CAD Formats (or FBX if so desired). Then on in the Export CAR Formats dialog box, click the Options button, and then in the Export Options dialog box, choose Export as ACIS solids. Click OK to close and then click Save to create the file as currently placed and named.

Chapter 15 Revit Structure Analysis

Configure Revit Structure structural settings and create loads for your project. The Structural Settings dialog box contains the tabs that will allow you to configure loads for your project. Load Cases, Load Natures, Load Combinations, and their usage form the basis for preparing your analytical model for export to analysis software.

Master it

1. True or False: Revit Structure has the ability to perform structural analysis.
2. True or False: Revit Structure cannot combine load cases.
3. True or False: A circular reference will occur when a system of beams frames back to the origin.
4. True or False: A good example of a typical boundary condition is the support of earth underneath a footing or a slab on grade.
5. Where are the settings located that allow you to turn on consistency checking?

Solution

1. False. The analytical model must be exported to analysis software.
2. False. Revit can combine load cases.
3. True.
4. True.
5. The settings are located in Structural Settings dialog box, in the Analytical Model Settings tab.

Place analytical load patterns onto your model. Loads are placed in the model in anticipation of using them for preliminary analysis. Several placement methods are possible within Revit Structure. Each of these methods can be applied in two ways.

Master it

1. Name the three different kinds of load placements.
2. The analytical properties of an element can depend on one of two things. What are they?
3. True or False: The directional guide is an icon that indicates the work plane for the load you are about to place.
4. What two ways can load placements be applied?
5. True or False: Adding a line load with host is a little less confusing than adding an area load since a line load has no z direction whatsoever.

Solution

1. The three kinds of load placement are line, point, and area loads.
2. The analytical model is dependent upon its host geometry, or it can be configured to extend to other members regardless of its host geometry.
3. True.
4. The load placements can be applied by clicking in the display area, or you can choose a host, resulting in six different ways to place a load.
5. True.

Import and export your virtual model from Revit Structure to structural analysis software. Once the loading is created and placed, the model is ready to be exported to an analysis application. Once the analysis is complete, it can then be imported back in to Revit Structure and will automatically update the model.

Master It

1. True or False: The integration links come prepackaged with Revit Structure.
2. When the application's dialog box is displayed, you will be given some choices. To what do they refer?
3. What type of file will Revit Structure export to the analysis application?
4. True or False: Importing and exporting to the analysis application exchanges the physical model back and forth between Revit Structure and the analysis application.

Solution

1. False. You have to go to the vendor's website and download the integration links and then install them.
2. They refer to the extent of elements that you are exporting and specific application questions you will need to answer for that particular analysis package.
3. Revit Structure will either export a file in that program's native extension or create a `.bim` file.
4. False. Only analysis data flows between the two applications.

Chapter 16: Project Phases and Design Options

Create project phases to manage element assignments. Creating and managing phases in a project is an important task that will help establish the sequence of construction of your structure. Phases apply existing and new statuses on elements so you can manage them. A good example of using phases is in distinguishing existing elements from new when you add a wing onto a hospital complex.

Master It What steps do you take in developing phases for your building document set?

Solution First, you have to either use the existing phases or create new ones. Then, assign elements in your model to the different phases you have created. Finally, create views that display the phases the way the project requires. Be sure to pay close attention to your element and view properties when you are working with phases.

Display project phases in your project views. Views in Revit Structure are configured to display your various phases. Using the Phase and Phase Filter parameters, you set each view to display new, demo, or existing objects, or any number of construction sequence views.

Master It The phase is set to New Construction, with only the Existing phase preceding it. Describe what each of the following phase filters will show:

1. All elements from the phase before New Construction
2. Demolished elements and all of the new elements that you add to your model

3. Only the new elements that you have added to your model
4. All elements, including existing, demolished, and temporary elements
5. All elements from the previous phase that were not demolished, as well as all added new elements

Solution

1. Show Previous Phase
2. Show Demo + New
3. Show New
4. Show All
5. Show Previous + New

Create design options to manage element assignments. In a design situation, you have to create sets of design options in order to evaluate various issues and problem areas. All of these options are created and managed in one Revit Structure file. They are then displayed in various views that you create.

Master It Answer the following questions:

1. How many options from an option set can be shown in one view?
2. What elements are not supported in design options?
3. What is a dedicated view?
4. How does using design options in your projects save you time and energy?

Solution

1. No matter how many options are added to an option set, only one can be displayed in a view at a time, which is called the primary option.
2. Levels, views, and annotations and 2D details are not supported in design options.
3. Dedicated views are developed to display one particular design option.
4. Using design options in your projects saves you considerable time and energy because you can display multiple design ideas easily without duplicating your project files.

Display design options in your project views. Once design options are created in your project, you assign them to different views. Those views are then added to a sheet for comparison. In that way, you are able to evaluate and select primary options and discard ones you do not want as the design process progresses.

Master It Some modeled elements in your project depend on other elements and so cannot be independently assigned to a design option. Name three of those types of elements.

Solution Elements of this sort that are dependent on their host objects include the following:

- Inserts that cut their hosts
- Host sweeps and their hosts
- Curtain panels
- Window mullions
- Grids
- Topographical surfaces and building pads

Also, elements that are added to a group must be in the same design option as the group.

Chapter 17: Standards: Increasing Revit Productivity

Interpret what can and cannot be done easily. Standards are there no matter what. What they are and how they are controlled are up to you. Before Revit Structure, users had various files provided with AutoCAD that determine what most of the standards were based on. A standard is many things, but a basic one specifies line weight, pattern, and style. Additionally, you have filled patterns that can tone contained areas. You then bring all these definitions together and apply them to object styles, views, and objects to create your drawings.

Master It Develop line weights that meet your needs. Create line patterns and styles and then apply them to your model via Object Styles. Address annotation and fill standards as well, all matching your required standards.

Solution Research pen weights to learn which are most applicable to your firm's standards. Examine how your line patterns look and adjust them to suit your needs. Do the same for fill patterns and the various annotation objects. Once ready, you then create your own line styles for use in your templates.

Enhance your model through customization. Anyone using Revit Structure deserves to have their tool be as productive as possible. With a little bit of practice, you can take using Revit Structure to a new level.

Master It Anyone can add a new Split with a Gap command, as well as control the size of temporary dimension values. Follow that with gaining access to your firm's library files as well. Then improve usability by using command shortcuts and take advantage of input speed.

Solution Locate and open your `Revit.INI` file for editing in Notepad. Then add an entry for creating the Split with a Gap command. Once complete, modify your `KeyboardShortcuts.txt` file to add one- or two-letter access to your favorite commands. From within Revit Structure, adjust your Options File Search paths to look to your custom family libraries first to enforce your standards automatically.

Implement model standards and view overrides. Once you have your standards in place, you then have to apply them to your model. There is no need to simply accept what Revit Structure can produce right out of the box.

Master It Take your standards and use them to control your model. Then when the need arises, tweak just about anything for a single view at a time. Can you break out of the black and white box and think in color again?

Solution After you have defined your standard containers, use objects styles to define them for the entire model. You can use the Visibility/Graphic Overrides dialog box to refine a single view. Then take a complete model and adjust the object styles to use color and notice how much easier it is to understand the objects in it.

Chapter 18: Family Creation: Beyond the Built-In Libraries

Create a footing step family. Creating families is a vital skill. You create a family, such as a footing step family, by meticulously tying reference planes with dimensions and parameters. Then you add 3D solids to the references to achieve a flexible, useful family.

Master It When you start a new Revit Structure family from a template, there will be existing reference planes. How do these reference planes help in the creation of the family?

Solution By offsetting these lines and dimensioning to them, you can establish a strong origin while maintaining flexibility within the family. These reference planes will also serve as alignment lines when you are modifying them in the model.

Create in-place families. You create specialized families, called in-place families, directly within the model, tying the family to the surrounding building. This allows you to more easily create custom geometry that will probably never be used in another model.

Master It What is the process for creating a custom family directly within the model?

Solution The first step in creating an in-place family is to choose the Modelling tab on the Design bar and click the Create button. Once you do this, the Design bar temporarily turns into the Family tab. From this point you can either create a solid 3D form or use any of the other choices from the Family tab.

Create groups. Adding groups to the model greatly reduces the time spent organizing and manipulating the configuration of certain items. Also, by linking a group, you can actually create a separate Revit Structure model and link it back into Revit Structure similarly to create an x-ref in AutoCAD.

Master It You also learned that a linked Revit Structure file can be turned into an embedded group. Explain the procedure for this to occur.

Solution Once you select the linked Revit Structure model, you can click Bind on the Options bar. From there you get a choice as to whether to include the attached details, levels, and grids. It is unusual to bind the levels and grids, as they create redundant instances, and Revit Structure will rename the levels and grids being imported.

Chapter 19: Advanced Structural Families

Create a parametrically driven tapered steel girder family. In this first section you learned how to develop a new family from scratch using the Beam template. You used a blended sweep to create the taper on the lower flange of the girder. You had to introduce a number of new reference planes to which you could then attach the sketch lines of the girder. Adding dimensions between reference planes and making them labels gave you the ability to flex the shape and made the family truly parametric.

Master It What does "flexing the model" mean, and why is it important?

Solution Flexing the model tests the parametric constraints of the model. You create several family types and then apply them to see if the shape reacts properly to the new parameters. It is important to flex the model in order to prove that the sketch lines that have been built into the shape are working correctly.

Construct an in-place bent beam family. The bent beam family was created inside the project as an example of an in-place family. You learned how to use a solid sweep form to sketch a path and a steel beam profile along a reference plane in order to create the desired shape.

Master It What members compose the bent beam family?

Solution The bent beam family consists of two members: the solid sweep form and the symbolic stick symbol.

Adapt the steel wide flange beam family by adding a nailer to its top. In this section you adapted the existing wide flange beam family by adding an extrusion onto its top. The extrusion represents the wood nailer and has the ability to flex into different shapes.

Master It What are the four main steps necessary to add an extrusion in the shape of a 3×6 to the wide flange beam family?

Solution

1. First you copy the family file to a file with a new name.
2. Then you copy its associated text file as well to the same name.
3. Finally you create a solid extruded form with a beam profile.
4. You add labeled dimensions to your reference planes in order to make the solid form act parametrically.

Create an elevator pit family that can be dropped into your project. You learned to make an elevator pit family that could be easily inserted into your project. The pit form consists of solid extrusions and void extrusions working together. The whole family is floor based, so it can exist only when associated with a floor object in your model.

Master It What do the void extrusions do in the elevator pit family?

Solution One void extrusion is locked to the top and bottom of the slab and the inside of the pit walls. This one automatically cuts the hole in the floor when the pit is added to the project.

The second void blend is used to cut out portions of the pit slab.

Produce wood and steel truss families using the truss template. In the final section you first worked through an exercise in which you inserted a truss from the structural truss library and then reconfigured it to create a steel truss with WT top and bottom chords and double-angle web members.

Then you used the truss template to create a wood truss from scratch so that its vertical web members could be specifically located above supporting walls in a warehouse.

Master It After you inserted the wood truss into your building model, how did you make the top chord of the truss follow the barrel-shaped roof?

Solution To make the top chord of the truss follow the barrel-shaped roof line, you first highlight the truss that you have inserted, and then on the Options bar you click the Attach/ Detach Top Chord button, and the top chord is then automatically reshaped. All web members are also extended.

Appendix B

The Gallery Up Close

Revit Structure is capable of modeling as well as creating documentation for a variety of building structures. To recognize what tools should be used and understand the various ways to create and document elements in a nonstandard structure, you need a good grasp of the techniques you've read about in this book.

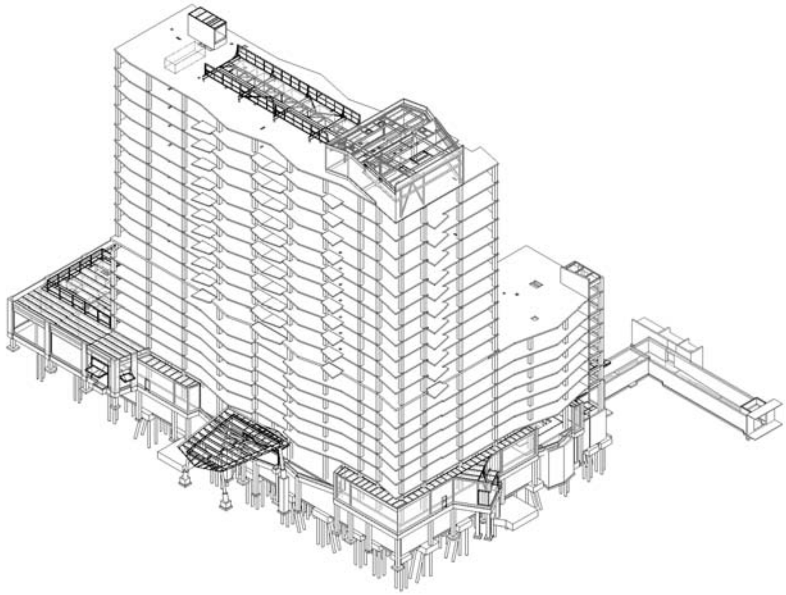
As you'll see in this appendix, the tools provided in Revit Structure to aid in these efforts are not always used for the purpose for which they were intended. As we discuss the wide range of projects we've modeled and the strategies we employed when using Revit Structure in a real-world production environment, you should gain a good understanding of how to overcome structure issues.

In this appendix, we'll show you examples of real-world projects that we've worked on using Revit Structure. We'll explain what we've done to overcome challenges and roadblocks, and discuss common situations that can be handled by stepping outside the box or taking a look from a different angle. We'll bring to light new workflow methods that you can experiment with to continue to push Revit Structure to new limits.

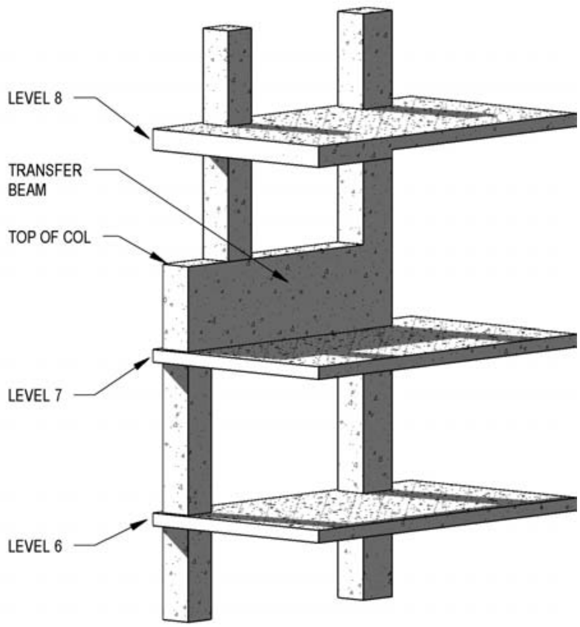
Mid-Rise with Multiple Structure Types

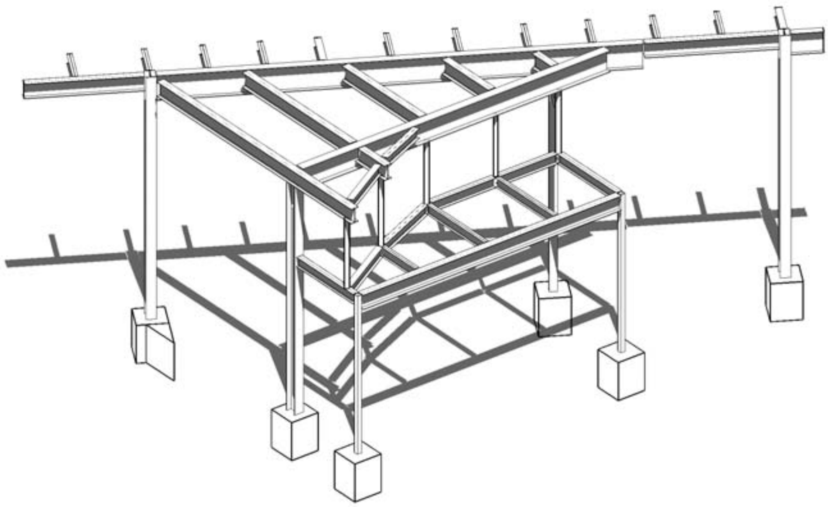
The hotel and residence shown in the following graphic is an 18-story structure with low-level ballroom and restaurant areas, as well as two levels of parking below grade. The main structure system is concrete, which supports the post-tension upper levels with concrete columns and shear walls. One level of supported parking below grade consists of a post-tension slab; the main street level is a two-way flat plate slab. Areas of the lower-level roofs are supported with a steel structural system consisting of columns, beams, and bar joist. The foundations used a mixture of conventional footings and pile caps, with piles and associated grade beams.

You can take several approaches when starting to model a project of this size as well as the various structural systems that it can use. Much depends on the schedule of the project and when the various portions need to be issued. For each project, regardless of its size, you must take into account its construction schedule, team member expectations, and limitations of the current software. Another factor is whether other design team disciplines are using BIM software or 2D CAD to produce their documentation.



For this project, we issued an early foundation package, so much of the work up-front involved designing the foundation. While doing this, we kept the upper portions of the model going, but only to exchange information back and forth with other disciplines to keep their design moving forward. For the upper portions of the structure, we focused on the modeling elements for design intent and not for documentation. The following images are examples of how we dispersed early design information to the team:





To help meet the quick construction schedules, we used strategies such as grouping floors for typical framing. For the foundation efforts, we were in full documentation mode, at first creating many 3D views to portray the design to the contractor and other design team members. We tagged everything with labels and used spot dimensions to call off elevations. This allowed us to keep the documentation up-to-date as the design progressed and the model was coordinated.

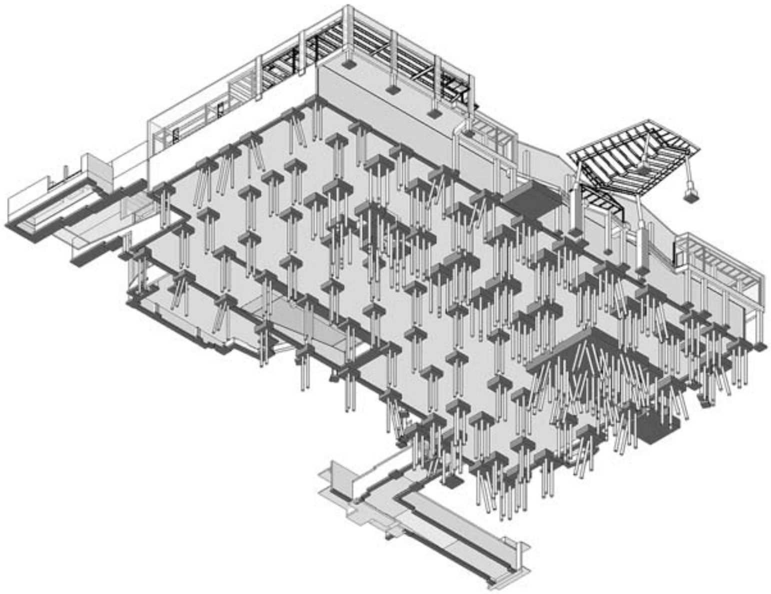
Modeling Techniques

Since several systems were used on this project, we employed various modeling techniques while working on our construction documents. Also, with projects of this size you can wind up using several versions of software—which in most cases will make some modeling tasks that much easier. This project started out in Revit Structure 3, and we just recently upgraded the project to Revit Structure 2009 for the last month or so of the construction phase. As we look back to day one, it would have been nice to start this project in Revit Structure 2009—we could have taken advantage of such improvements as warped slabs, more enhanced sloping beams, filters by criteria, dependent views, concrete joining, slab edge, foundation pads, enhanced spot dimensions, schedule columns offset from grids... As new features are added, you have to be ready to adapt to them or make the decision to carry out your original modeling techniques. Many of the techniques used on this project were workarounds for solutions that are now much easier to accomplish in the current release. Even though some of these work-around techniques have been replaced with added functionality, some are still methods that are preferred or at least another option that can be used for special situations in just about any project you come across.

Pile Foundations

For the pile foundations, we used the Pile Cap families that shipped with Revit Structure. Not all shapes and pile cap types were available, so we created additional ones as needed, using the existing ones to start from. The structural design required that several of the piles had to be

battered, which meant we had to create separate families for the various locations to accurately show the location of the battered piles. Since the cap families nest in a shared pile family, we were able to schedule the individual piles, which allowed us to keep an up-to-date count of the total number of piles at the various design stages of the project. We assigned additional parameters to the families to include more information to be scheduled, such as bearing capacity and whether the piles were battered. As you can see in the following view, showing the foundations providing support enhanced the visual aspect and put the design in perspective for those viewing it.

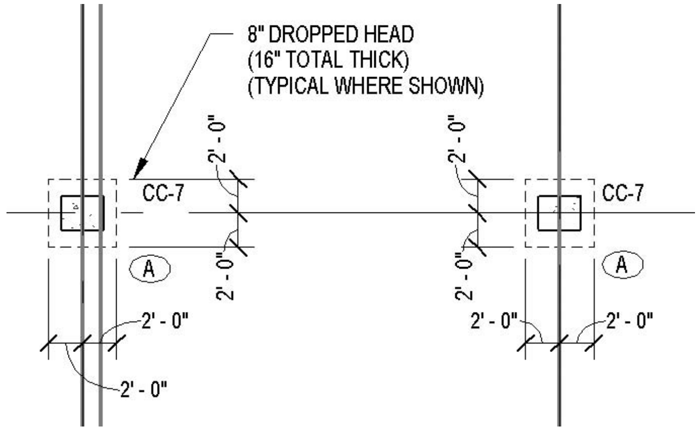


Columns

Concrete columns were modeled with the modeling as it is built approach. They were modeled individually floor to floor, with their tops attached to the underside of the slab or column capitals. We created additional text parameters in the properties of the concrete column families for reinforcing so column reinforcing could be scheduled. Because the columns were modeled floor to floor, we were able to put all column reinforcing information into each object in text form. Aside from creating construction documents, this information was also displayed in views via tags for checking the integrity of the model as well as more efficient checking of shop drawings.

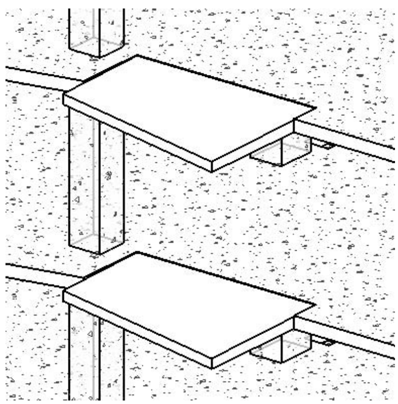
Slabs

We used the slab tools to model the slabs as well as any dropped slabs or column capitals. Using the slab tools for these elements gave us more flexibility in modeling the various shapes and sizes that were needed throughout the project. For instance, if an 8″ slab required an 8″ dropped slab, we'd create a new slab type called Dropped Slab – 8″. This slab would be placed with the same reference as the 8″ slab with an offset of –8″. With the slabs named to reflect their use, we created schedules for calling out reinforcing and for calculating concrete volumes.

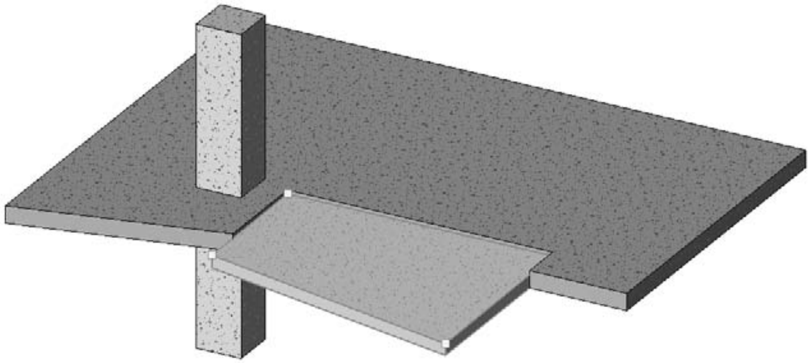


We decided to make the slab thickness the depth of the dropped slab and not the overall thickness so that we did not have to rely on the concrete joining behavior. With this method, the dropped slab displays as a hidden line regardless of its join behavior. This is only one strategy that can be used for modeling dropped slabs. Another method is to create a slab type that is the overall thickness of the slab at the capital. The two slabs would automatically join together, cutting out the areas of overlapped concrete. Using this method would eliminate having to maintain an offset for the dropped slab, but in return will not accurately reflect the total volume of concrete in the slabs.

Several of the upper levels had recessed balconies (shown in the following graphic) that were at the exterior of the building. With the Revit Structure release at the time we modeled these recesses, and we used an in-place family that existed for each level. Each family contained individual void blends to form the recess. We used a void blend sketched in a section view because the balcony was sloping and each corner of the recess was at a different elevation. We placed reference planes inside the family over the top of the recessed lines to give us the ability to dimension to the recessed edges. Without adding these reference planes, we would not have been able to select the recess edge as a dimension reference point.

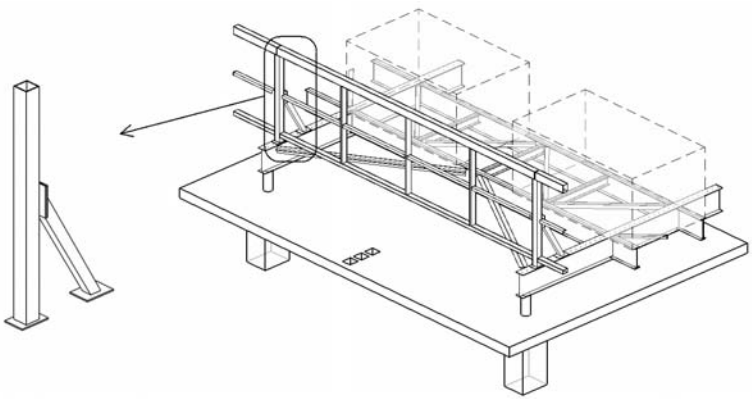


With the current version of Revit Structure, modeling recessed slabs like this is much easier. One method involves using the slab tools in Revit Structure. You model the core slab around the recessed slabs and then come back with individual slabs at each recess. With this method, you can treat each recess as its own slab, thus giving you the ability to control the various elevations of the recess slab by using the elevation tools located on the Options bar when you select the slab. When complete, all of the slab types will be automatically joined to show one monolithic slab. You will also need to use the Variable option in the Edit Assembly dialog box for the slab. Setting the Variable option will keep the bottom of the slab flat rather than sloping with the top surface.



Screen Wall Framing

Screen walls (shown here) are often required on structures to mainly conceal mechanical equipment. These types of structures can typically be modeled by using structural columns and beams.

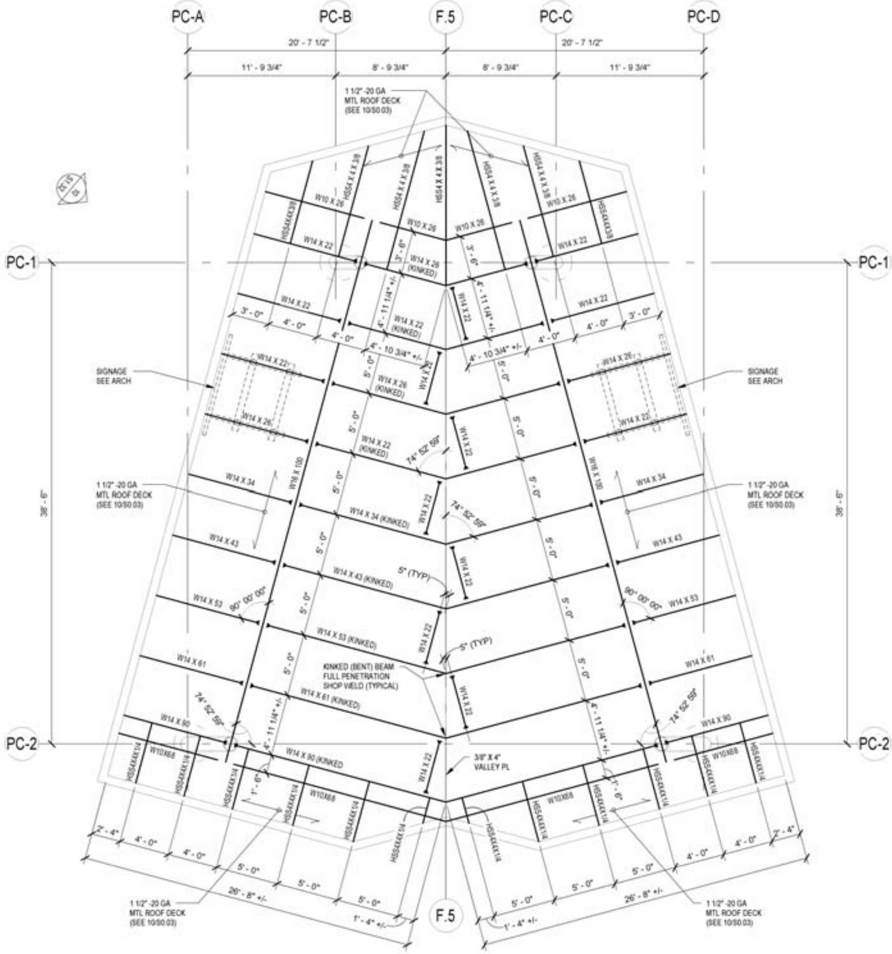


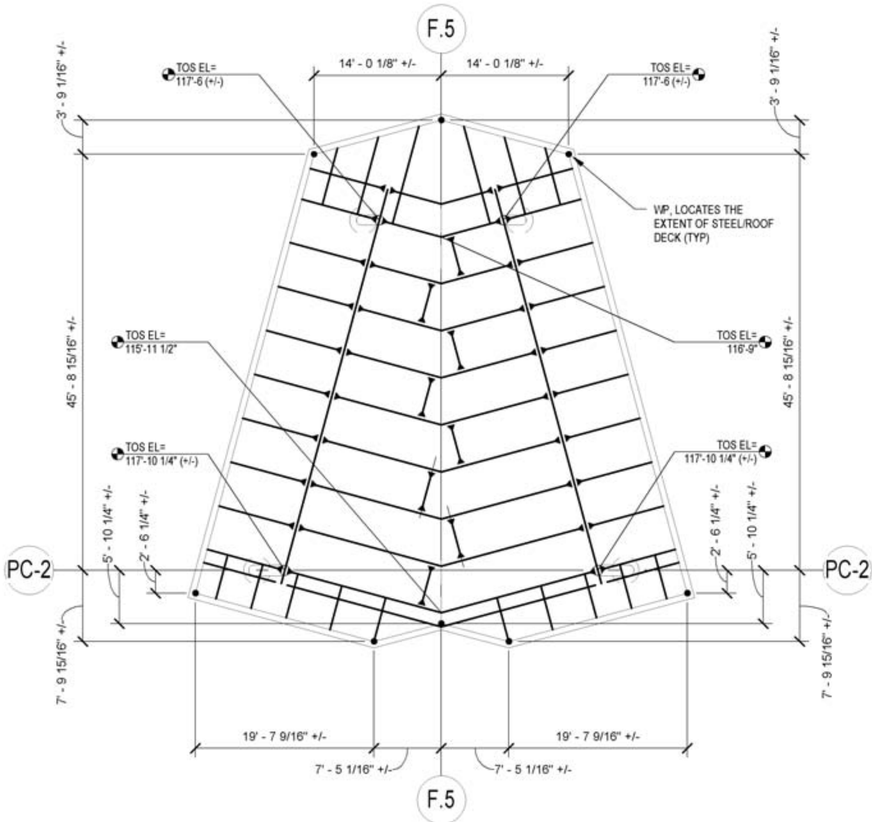
The vertical supports and its kickers were all typical at the perimeter, so for this project we created a column family that we used specifically for the screen wall. We used Revit Structure's HSS Tube Steel Column family as the template. In our new family, we reassigned the base constraints of the column as well as added new geometry for the plates and kicker brace by using extrusions. Once loaded back into the project, all locations where this framing was required instantly updated. We assigned the new geometry visibility settings to allow the proper display

depending on the type of view. Modeling to this detail helped us to detect conflicts with the mechanical units as well as ensure that any clear space was not being encroached upon. As the location and slope of the kickers were revised to accommodate the ever-changing mechanical units, this method helped us make those changes quickly.

Sloped Canopy

This project also had a unique canopy structure. Without Revit Structure, we would probably *still* be trying to figure out how it was all going to come together in regard to its connection requirements and relationship with the architectural skin. For this project, we chose to model the canopy as its own model. This allowed us to have several design option models to choose from. Each design was easier to modify and maintain as a separate model. This approach also enabled us to link the entire structure into each canopy model to show images of the various design concepts and how they related to the structure. In the end, the chosen canopy design concept was linked back into the master Revit Structure model, where we added it to the documentation set. At this point, we were able to create plans and sections to facilitate documenting the model.

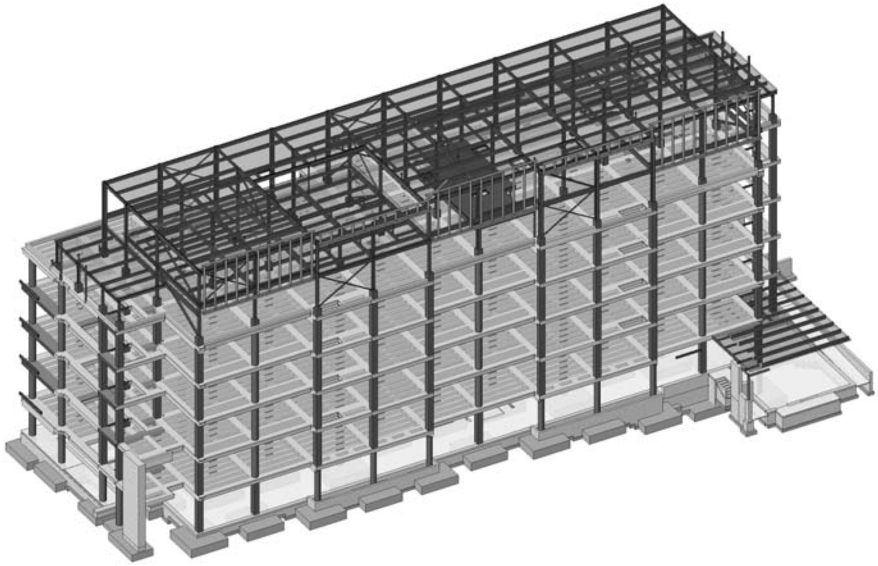




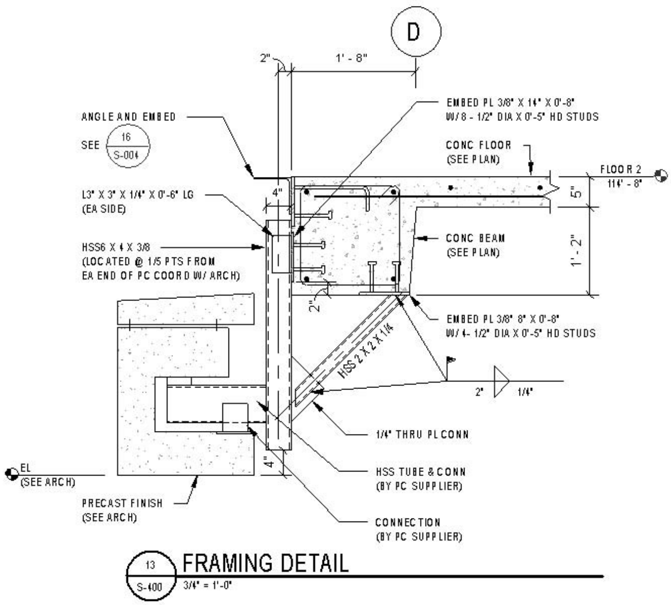
With Revit Structure's ability to create multiple views of the model at different scales, we were able to create separate plans to dimension the location of the steel and for denoting top-of-steel elevations. This resulted in plans that were clear and to the point.

Concrete Pan and Joist Structures

Concrete pan and joist structures (like the ones shown here as well as on the color insert pages) may require more expertise—or at least a few pointers—in using Revit Structure to achieve a good set of documentation while still keeping as much information in the model as you can. The last few releases of Revit Structure improved the program's capabilities with regard to concrete framing and how it joins. With the 2009 release, not only do the concrete framing members automatically join to other members of the same material, but you can also control the types of beam-to-beam join types, similar to how you can with a wall corner or intersection. This works with rectangular shapes, but in a pan and joist system the members are tapered on their sides, thus making the Edit Beam Joins tool unusable. For a pan and joist system, you will have to develop other methods to overcome some of the limitations of the software for the tapered shapes that this system requires.



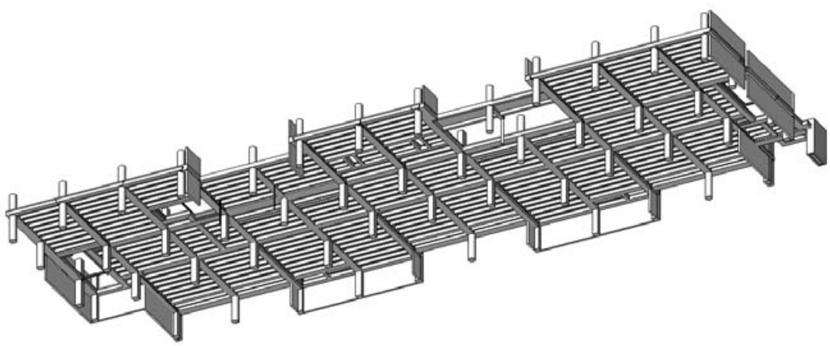
Before you start to model this type of system, make a list of everything you want to get out of your model. How will the model be used? Do you want the ability to cut and annotate live sections from the model, like the one shown here, or will live sections be created by using a drafting view?



Are you only using the model for geometry display, or will you be building in additional information that can be scheduled by using the properties of each individual element? If you are only using the model for geometry display, then you could probably just model each floor with a slab and cut extrusions. Or you could model each floor as one big in-place family, with several extrusions to form the pan and joist system. (Of course, with this method you will have zero intelligence in the model. All information denoting these members will be 2D text and it will be difficult to have Revit Structure automatically create a schedule.) Will the model be used for collaboration by another discipline? Do you want to calculate concrete volumes? Questions like these will help you determine the best approach to take before you start modeling a project with this structural system. We prefer to discuss the method that allows you to put as much information into the model as you can, accurately model the system, and easily extract accurate information out of the model to be displayed for your documentation.

What's in the Family

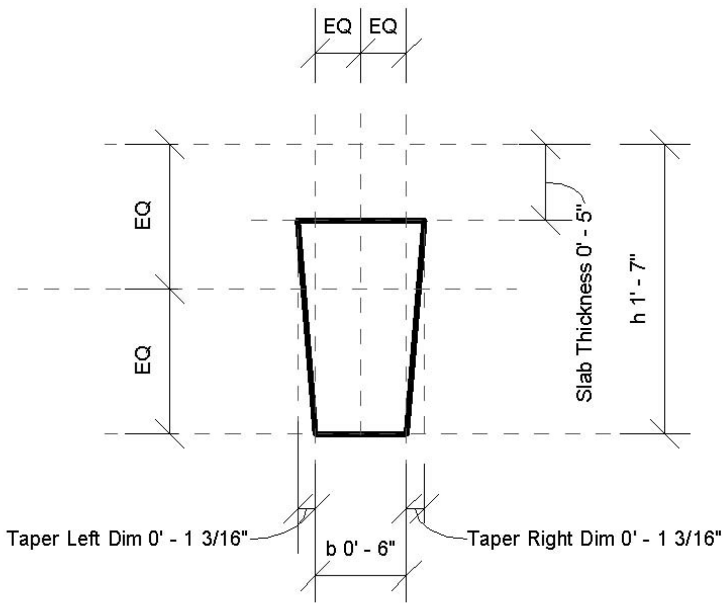
When Revit Structure was first released, it did not feature families that we could accurately use for a pan and joist system. We had to start from the basic rectangular beam family and develop our own families for the typical shapes that we required. At first, we had created shapes for a concrete tapered beam, a tapered beam with a shallow pan, a tapered joist, and a tapered bridging member. These all were just two basic shapes used in the pan and joist system. We created separate families for each to give us more flexibility in scheduling as well as assign color for the surface material so we could visually display the various structural components of the system. Families for these two basic shapes are located in Revit Structure's `Imperial Library\Structural\Framing\Concrete` folder. Even though some of these families are available to you after installing Revit Structure, it is important for you to realize that you will still have to develop certain families of your own and even project-specific families as you get further along into your modeling adventure. Just because everything is not available out of the box doesn't mean that modeling a pan and joist system cannot be done successfully.



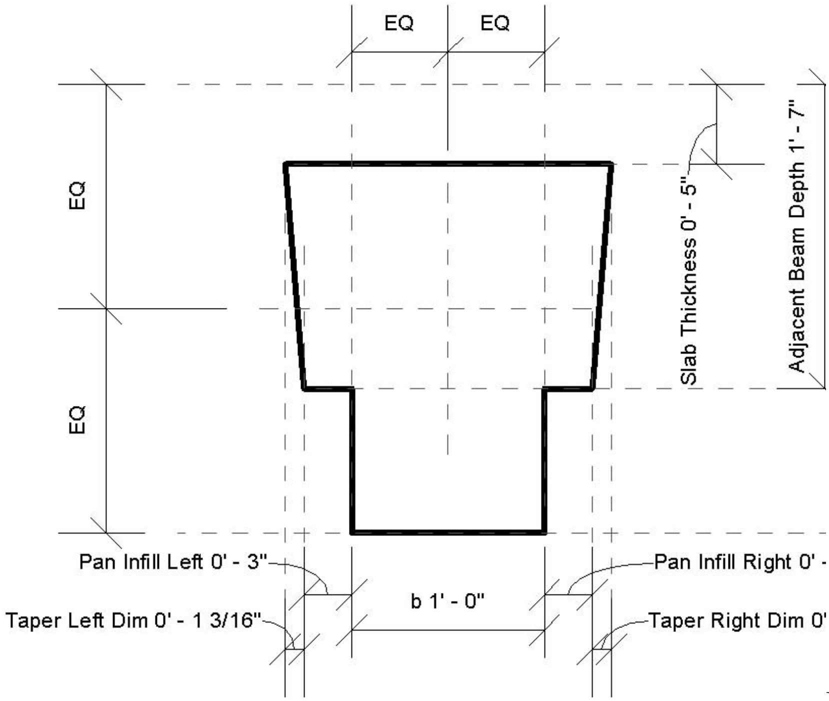
As you get further along in a concrete pan and joist structure, you will begin to see that you must create new family shapes to help model the structure accurately. You will also want to keep in mind that if every framing member needs to be scheduled as well as annotated based on the

member's properties, you must choose a modeling method that will allow you to do that. This is where you will want to get your in-house family creator expert involved. Many families can be created quickly by starting from a current family, or similar ones can be used by adding more advanced functionality to them. You will have to create many families specifically for a project. The more projects you do, the more families you may break out from being project-specific and add them to your standard family library.

The basic tapered beam family shown next has parameters that help control the geometry that are different for controlling the geometry of a rectangular beam. The first are the parameters for the Taper Left Dim and Taper Right Dim. These parameters are driven by a Yes/No setting as properties of the tapered beam and have formulas assigned to them. When the user chooses, Revit Structure will display the edges of the beam tapered (yes) or show them straight (no), as in a rectangular beam. This allows you to use the tapered beam as an interior beam, with the beam tapered on both sides, or at an edge condition, where only one side is tapered. You must coordinate the Slab Thickness parameter with the slab thickness that the tapered beam is being placed under. This keeps the taper dimension accurately shown, automatically displays the bottom edge of the beams as hidden lines when the beam is not joined to the slab, and ensures accurate totals of volume counts for the slab and beams.



The next basic family shown is for the condition where a beam is deeper than the beam or joist adjacent to it. This type of beam is only tapered for the depth of the adjacent beams and has vertical faces for the deeper portions of the beam.



]

The same parameters exist for this family as the one described earlier. Additional parameters have been added to this family to control the Pan Infill Left and Pan Infill Right, as well as the Adjacent Beam Depth. To achieve the proper display of line work on plan, you need to always have these options set correctly. In order for an adjacent beam to join and display properly, the Adjacent Beam Depth parameter must be set to the proper depth. Setting any of the Pan Infill parameters to zero along with turning of the taper will create an edge beam condition. This family could be taken one step further to add another level of functionality by expanding the Adjacent Beam Depth parameter so you set it differently for a Left or a Right condition.

There is also a parameter setting in the family called Display in Hidden Views that defines how the hidden lines of concrete behave in the project. This parameter will become available only when the family is set to a Concrete material. For the families mentioned earlier, this option is set to Edges Hidden by Beam Itself. This setting allows the bottom of the tapered concrete beam line to display as a hidden line rather than where the taper meets the slab.

Along with creating the geometry of the family, you will want to add in text parameters to give you the ability to put all of the reinforcing information into each element you create. Make a list of required parameters that matches your methods of scheduling. Once you have this list, add the parameters to your shared parameter text file. Each family that you create—as well as those families that are created in-place—should have these shared parameters incorporated into them. For families that are created in-place, the shared parameters can be added as project

parameters. Developing your families to hold text such as reinforcing information will allow you set up Revit Structure to automatically create and maintain your schedules, as shown here:

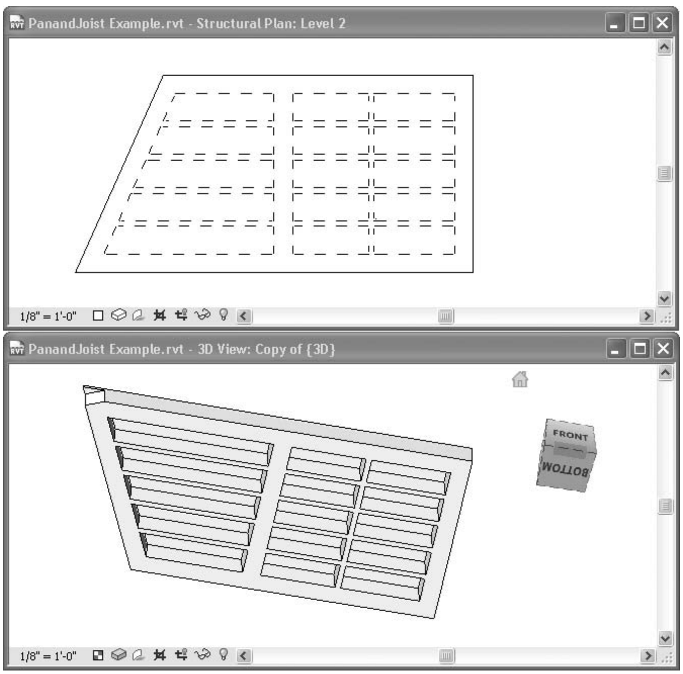
CONCRETE BEAM SCHEDULE - LEVEL 2

BEAM MARK NUMBER	PLAN SIZE	BEAM SIZE		TOP BARS (LEFT)	BOTTOM BARS (CENTER)	TOP BARS (RIGHT)	STIRRUPS			REMARKS
		WIDTH	DEPTH				LEFT	RIGHT	TYPE	
B2-001	35 X 36	35"	36"	10-#9	5-#9	10-#9	12	12	S	TOP BARS CONTINUOUS
B2-002	35/43 X 19	35"	19"		5-#7	5-#8	8	8	S	TOP BARS CONTINUOUS (REF DETAIL 12/S004)
B2-003	43 x 19	43"	19"		5-#6	5-#6	8	8	S	
B2-004	26/37 X 19	26"	19"	5-#7	5-#7		8	8	S	TOP BARS CONTINUOUS (REF DETAIL 12/S004)
B2-005	30 X 24	30"	24"	5-#9	5-#9	5-#8	10	10	S	
B2-006	30 X 19	30"	19"		5-#7	6-#9	8	8	S	

Workaround Solutions

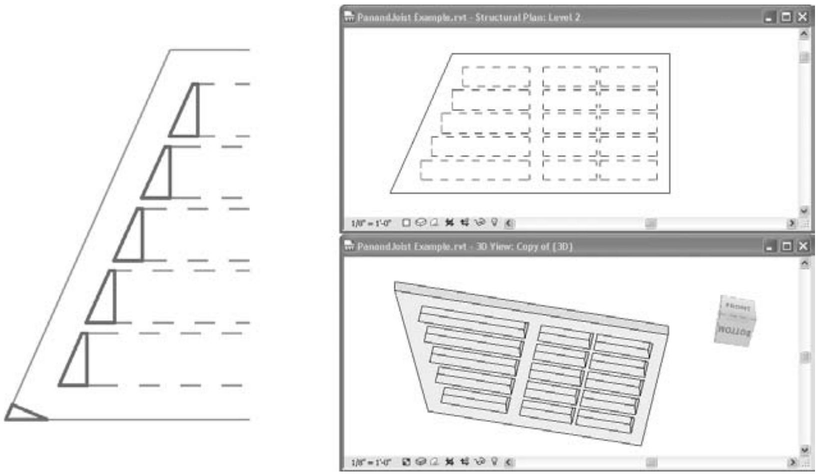
So, what do you do when a beam does not join as well as you hoped it would, or the line work shown on plan is not displaying correctly? You can take advantage of solutions such as using the Line Work tool, opening by face, or placing a slab as an in-fill slab to either finish what you couldn't accomplish with the Revit Structure tapered beam capabilities or to just put the finishing touches on your documentation. Again, try using the version of Revit Structure you have and the families that come with it—then decide whether you need to invest in creating custom families or workarounds.

In the following image, the plan view displays line work correctly for the condition at a beam-to-beam join at a sharp corner. However, the accuracy of the model is not correct. Also note that the form work for a pan and joist system comes with 90-degree ends. The form work pans will stop short of the beam, leaving a triangular area filled with concrete. You can address problem areas such as the corner beam intersection and the triangular in-fills by using the slab tool.



Create a slab type and call it something like **Slab Infill – 24″**. It should reflect the depth of the beam for the pan and joist system or the depth of the beam minus the slab thickness. Use this slab type to add the additional geometry that we mentioned earlier. If you are using the Slab Thickness parameter in the beam families, consider making this slab type the thickness of the overall beam depth minus the slab thickness. You would then offset the in-fill slab by the thickness of the slab. Use the same method for setting up your tapered beam families to help maintain consistency in your model.

In the following image, on the left is a slab in Sketch mode with multiple closed-loop sketch lines to model the in-fill areas. These in-fill slabs can be modeled as separate slabs or several individual slabs all in one slab sketch. You will need to decide which method is best for you. These in-fill slabs should automatically join to the slab and beam members as long as the materials are the same. On the right is both a plan view and a below view, indicating that the model is now shown accurately for both model coordination as well as documentation.



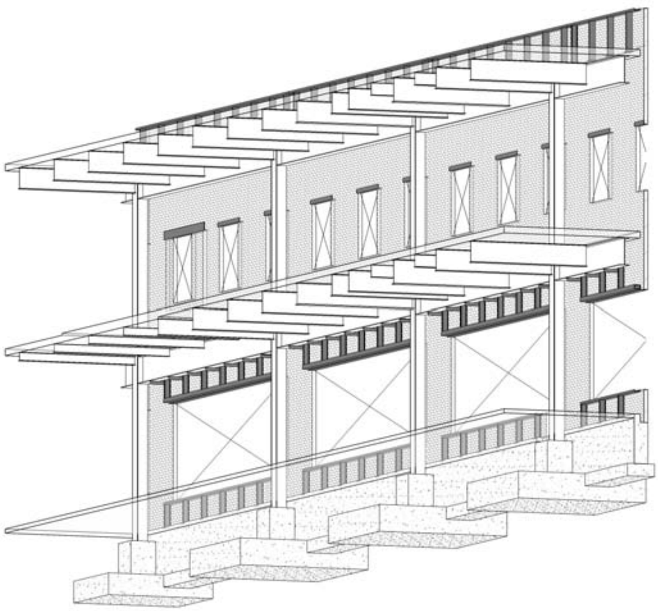
Other techniques, such as opening by face, can be used to cope or carve out tapered portions of the beam at openings or oddball shapes of beams. Using this approach may be a bit handier than creating a whole new family for a specific condition. You first select the plane of the element through which you will be placing an opening. Then place sketch lines to form a closed loop in the shape needed to create the look you want. This method not only should create the desired look in plan but should also be reflected in the 3D content of the model.

Tools such as the Linework and Edit Cut Profile tools (which are located in the Tools menu as well as on the toolbars) can help you fix unsatisfying areas of your documentation where it is not necessary to reflect the 3D intent throughout the model. Both of these tools are considered 2D or view-specific tools. This means that you may get the look you desire in a particular view but it will not be reflected in all views of the model. The Linework tool can be used to change the line type of lines that are not displaying correctly or to selectively remove lines that should not be displayed. The Edit Cut Profile tool is useful in sections where you need to change the configuration of an element. You select the object you wish to change the shape of and add sketch lines to remove or add to the shape in a 2D manner. The new sketch boundary automatically takes on the selected object’s material.

As you can see, a lot can go into a pan and joist structural system when you are modeling it in Revit Structure. Taking the time to discuss what you want, or what others hope to get out of the model, will certainly help you determine the best modeling techniques.

Projects with Miscellaneous Framing

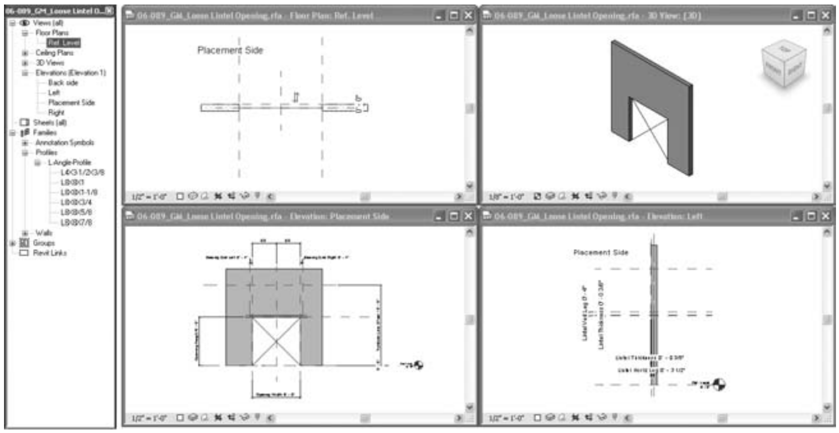
Sometimes taking a step back to see what you want to get out of your model in the end might just help you get to that finish line faster. Some methods may not work for every project—which is why you need to spend the time up front with fellow in-house team members and other design team associates who will be working on the project to discuss what you want to put into the model.



For this two-story medical building, we tried a few things that we would never think of attempting on some of our larger projects. We created all-in-one Opening/Lintel families and Brick Support Frame families. Why wouldn't we attempt that on larger projects? Well, if we tried to model to this degree and then expected to maintain the accuracy as things changed, or had to deal with possible performance issues, we might still be working weekends to make up for lost time. During our assessment of this project, we felt that enough of the design was decided, which meant things were not going to change (or at least *shouldn't* change). The schedule was pretty well set, and the model was small enough that we felt we could easily maintain it. What did we hope to get out of the project? Our goal was to develop schedules that linked to the modeled geometry, maintain accuracy, be able to share the accuracy with the clients to help coordinate with their sections, and make an attempt at scheduling quantities.

Loose Lintel Opening Family

We started the Loose Lintel Opening family from the Generic Model wall-based family template. We used a cut extrusion to cut the opening in the wall. We used model lines to display the "X" through the opening cut. We added reference planes and parameters to control the left and right bearing lengths. In addition, we created shared parameters which allowed the lintel properties to be linked into schedules. For the loose lintel, we used a solid sweep with an angle profile assigned to it so we could swap out the angle sizes that we needed for the various opening widths. We set the visibility parameters of the model lines and geometry so they would display in a specific way depending on the detail level of the view and whether it was a 3D, plan, or section view.



Once the family was working correctly with all of the desired functionality, we loaded it into the family and everything worked as expected. When we started looking at the plan, we said to ourselves, "Wouldn't it be nice to be able to tag and schedule all of these lintels we just placed?" Well, that was our next step. We figured that taking another hour or so building in this functionality might save us some time down the road. If it didn't save us time on this project, it would certainly save us time on the next one.

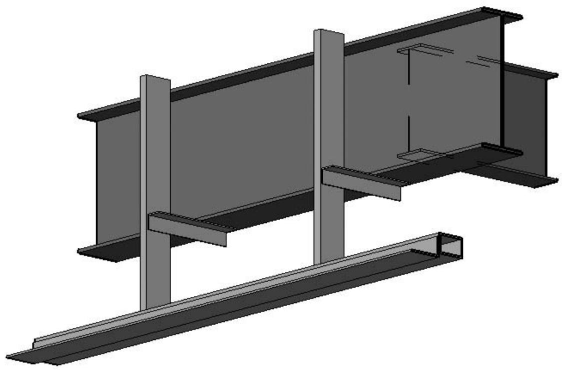
We added a symbolic line with its ends locked to the reference planes that controlled the bearing lengths. This allowed the symbolic line to accurately display the correct bearing length. Since openings can be placed at various heights, we added a parameter for floor-to-floor height so each opening we placed could control the vertical elevation of the symbolic line and thus would be properly placed in the view range that it needed to be shown in. Because the symbolic line was part of the family, we could tag it just like any other element in Revit Structure. At this point we had a lintel family that modeled the geometry accurately, its behavior was automated depending on the settings it was being viewed in, and its properties could be scheduled. Creating this family for this project has opened our minds (and hopefully yours) to other ideas with similar workflow methods. Taking this approach for this type of project proved to be an enhancement to our workflow.

Brick Support Frame Families

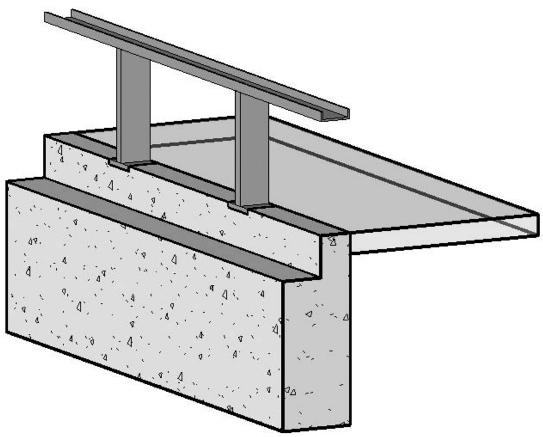
We also created families that consisted of channel frames with light-gauge stud in-fill. These frames were required at the top of some of the larger openings in addition to being used for

their sills. These families consisted of vertical and horizontal channel families created from solid sweeps that were nested into the master family. We then placed these nested channel families in an array parameter that allowed us to adjust the channel spacing to the length of the span. The basic setup was the same for both family types. Nesting the individual families into one another gave us more control over the behavior of the arrays that we used, and it also enabled us to create Family Type parameters to swap out channel shapes with HSS tube steel shapes.

The Brick Support Frame family shown here had a horizontal kicker angle that we were able to attach to the bottom of beams in the project by aligning and locking a reference plane set to an instance parameter. When beam sizes changed, the kicker brace changed with it. This not only put additional information into our model for a more accurate set of details, but it also helped the architect visualize and coordinate the extra structure required for the design.



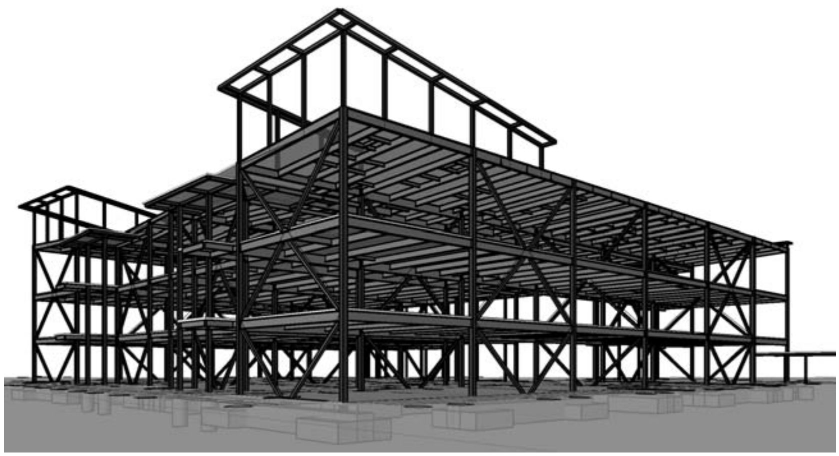
The Support Fame shown next also nested in an embed plate family. This did not add much benefit to the project, but the plate was automatically displayed in our section views and also made our 3D views a bit more attractive. We were having fun and things were going good—which made it hard to know when to stop.



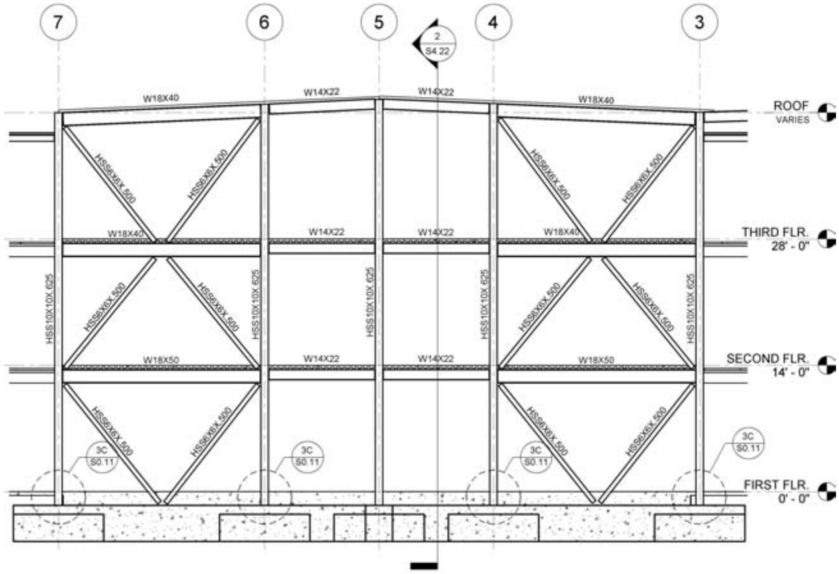
Once you gain a better understanding of how families work, you will find that you can build more functionality into your projects. Seeing all of these frames while viewing the project can be a bit overwhelming, but when you take a closer look, all of the information exists in one spot: the family. If changes need to be made, you make those changes in the family or by using the parameter functionality built into it. All those locations will update to your changes. As you learn to work with families and parameters types, you will be able to allow your model to maintain itself.

A Campus Design

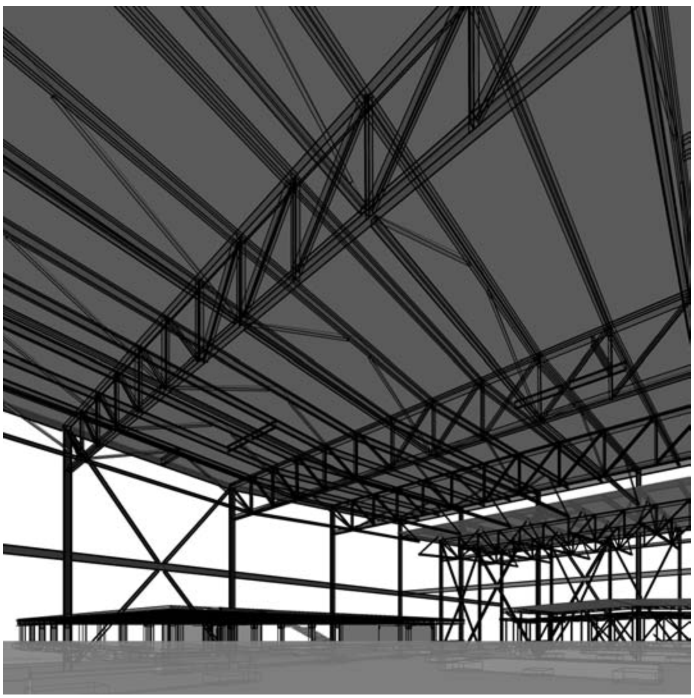
Consider how you might approach the modeling setup for a campus with seven structures: two multi-story classrooms, a library, a food services building, gymnasium, Administration building and a parking structure. Will you model all seven buildings in one Revit Structure file? Probably not. If you could, you might use worksets to divvy up all the buildings into discrete parts. For a project of this size, though, you will have several people handling the modeling and the documentation who will need constant access to the file. The engineers will also be involved in studying the model and possibly taking advantage of the analytical model. That could make for a lot of performance issues. Even though Revit Structure files tend to be smaller than their architectural counterparts, which range from 17MB to 32MB each, putting all seven in one file would make the file nearly 200MB.

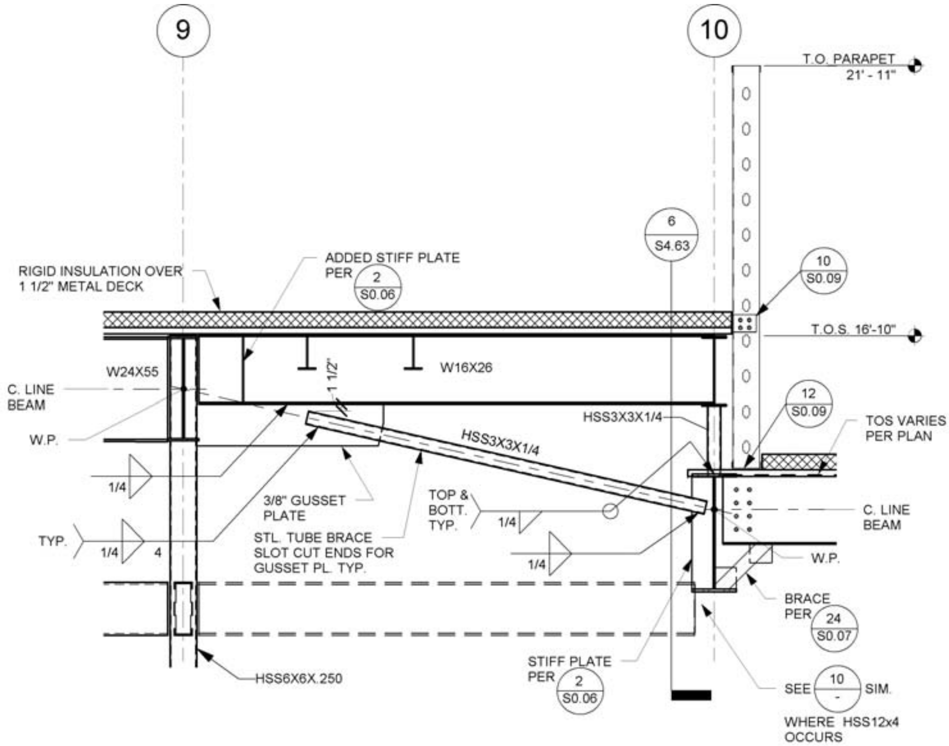


A better approach is to model each building separately; make each structure its own central file. By setting up the project this way, you ensure that several people can work easily in each file. Each file can have its own set of worksets. That is the approach taken by the team who worked on the LAUSD Central High School campus project. This approach can work for any campus-style design project.



The toughest decision was how to work on two classroom buildings that were basically the same, except for a few areas. It was that "except" that swayed the decision to make two separate models in separate files. So many times in real practice when you are told that buildings are identical, they really are not—and the frustration level can really build.



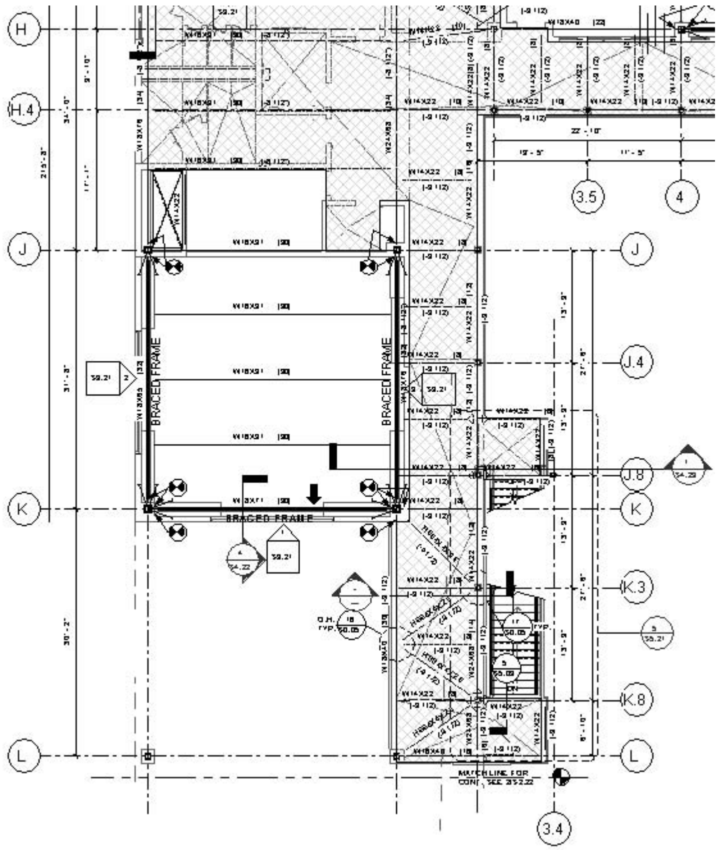


We decided to create an overall site file. When we created the individual files, we first inserted the architectural 2D site plan and each new building was located at its correct position within the campus. We then linked each building model into the overall site plan file using the Origin-to-Origin setting so that they all appeared in their correct position.

Each of the seven files contained sheet files for plan, elevation, and section work for that particular structure. So one downside is that plotting required opening all of the Revit files and involved many organizational duties.

The other major concern was where to locate the Typical Detail sheets. Those ended up in the overall site file. The overall site plan also became a central file so a number of people could be working at the same time on the typical details. One nice benefit of placing the typical details in the overall site file involved the creation of a drawing list for the project. The drawing list schedule was able to extract all the sheet information from the seven linked files and compile them into one list. That was a great help in coordinating the set.

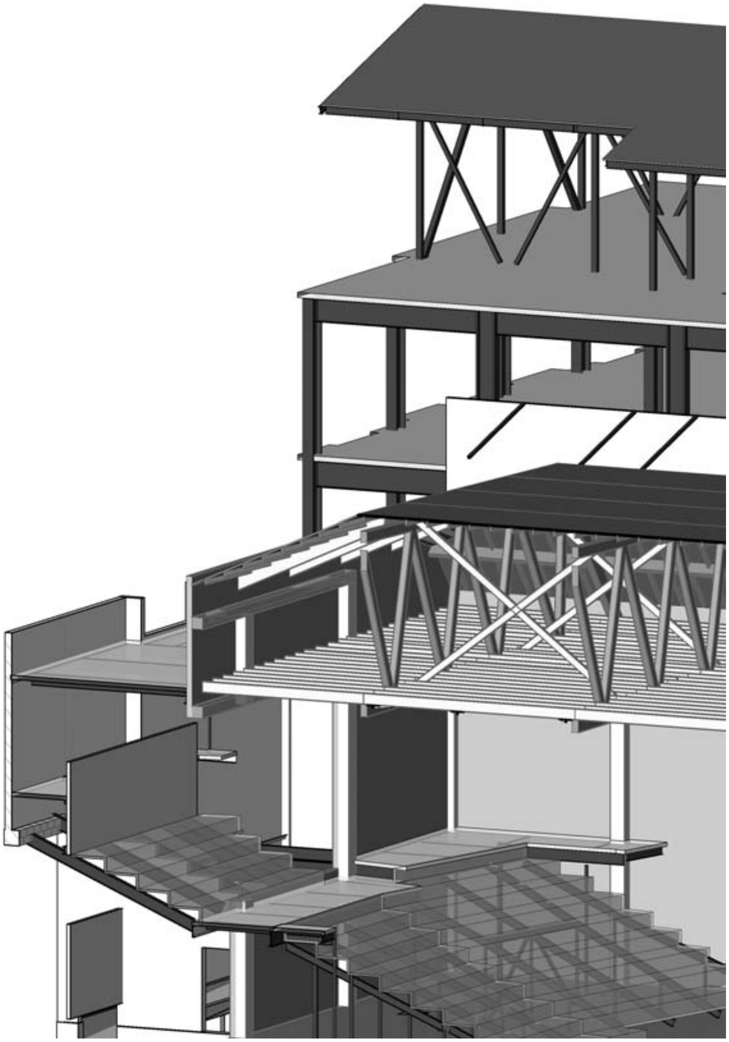
Besides the typical details, the site file contains several sheets of overall model views. You get a great feel for the whole campus when you take some perspective snapshots from the ground level looking across the campus. With products like Autodesk NavisWorks, you can easily cruise building to building through the whole site. For client presentations, this was the file we used.



A Hollywood Studio Project at NBC Universal

The NBC Live Show at Universal is to be the next home of the famous "Tonight Show." This was a high-profile project indeed. It had a demanding design and construction schedule since it had to be ready for the changing of the guard in 2009, as Jay Leno exits and Conan O'Brien takes over the show. The project had two main parts: retrofitting and remodeling the existing Stage 1, and adding a new support building.

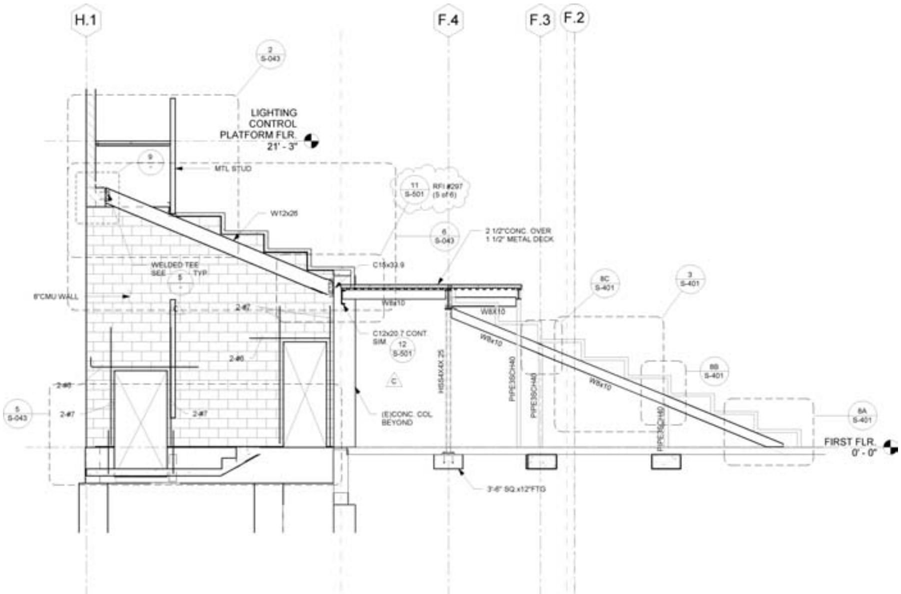
The existing stage in itself is famous. It was the first TV production stage built on the Universal lot back in the 1950s for the great comedian Jack Benny and his TV show. There is a lot of history at the Universal lot, and it was satisfying to work on a structure with such a storied past.



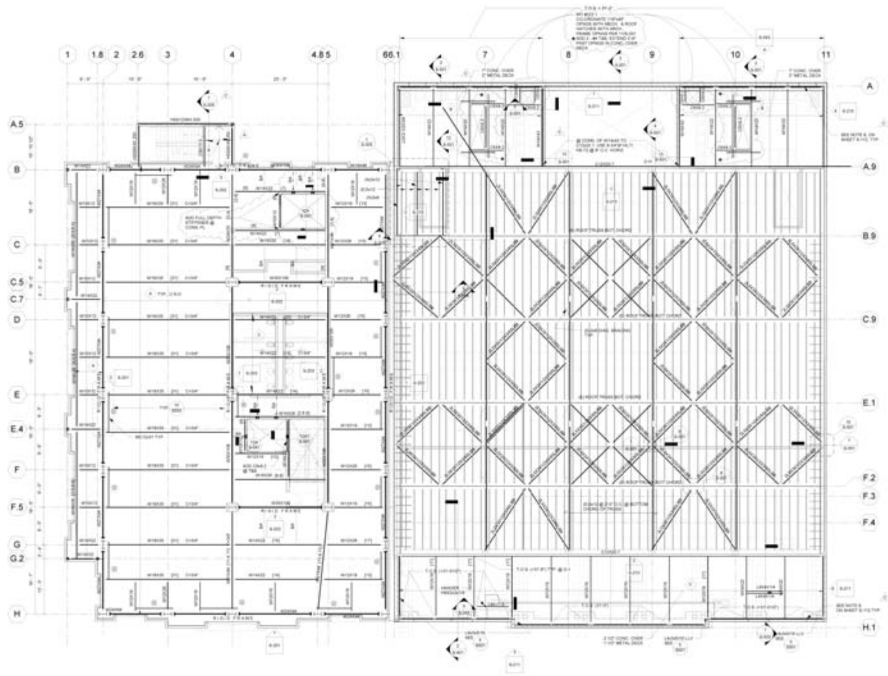
The stage was made of 6″ tilt-up concrete walls. Wood trusses carried the roof with lumber rafters and ceiling joists attached across the truss bottom chord. Wood cross bracing finished the existing roof design.

The new support building is a four-story steel and concrete structure with moment frames as the main seismic restraint system. There is also a full basement. The building has heavy Spanish roof tiles, so the roof needed to be strong to carry the weight as well as being braced diagonally on four sides.

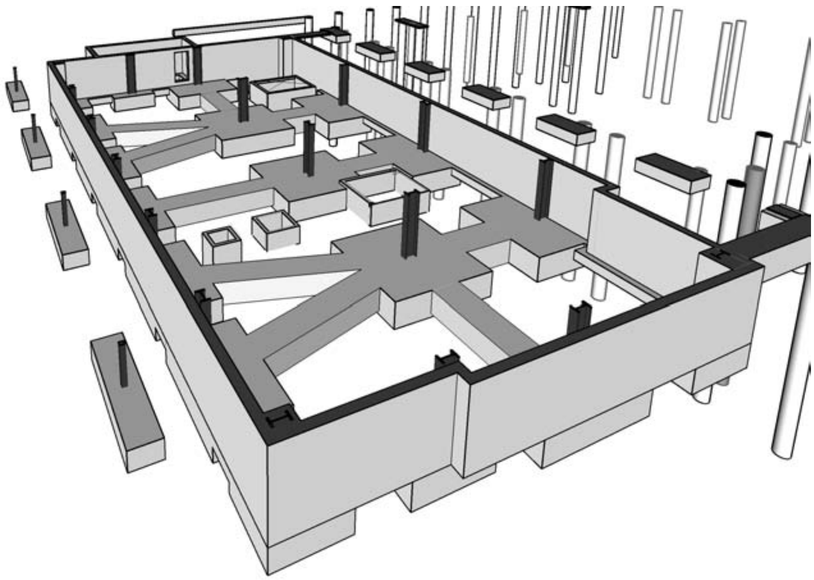
We added a new front entry and a back fly house (which holds all the sets). All new work in the stage area was done in the new Revit Structure file. We went back to the original stage file to demolish portions, such as the many new openings in the exterior tilt-up walls.



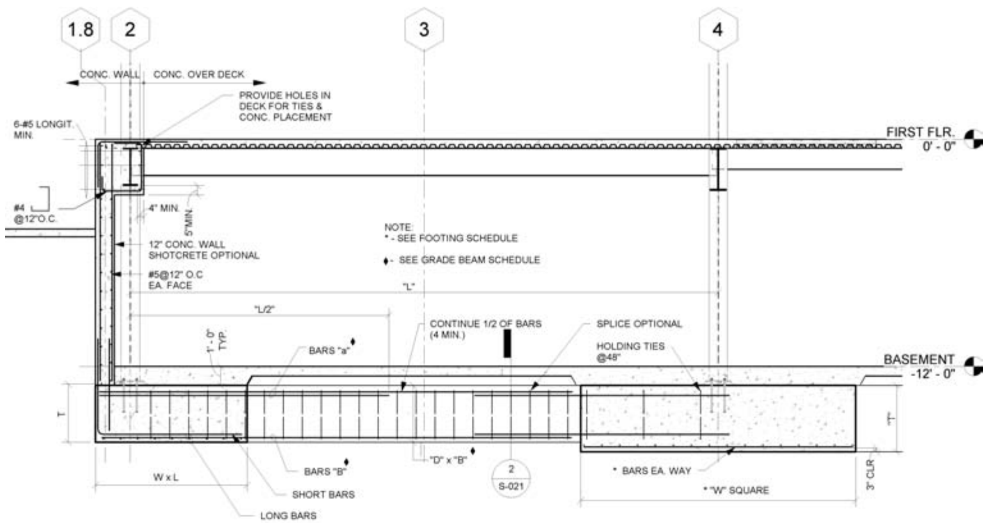
Our approach to modeling the project in Revit Structure involved the creation of two files: one for the existing stage, which was then linked to the second file containing the new support building. We set the Phase Created parameter for all elements in the stage area to Existing. The linked file was half-toned in each view to distinguish it from the new work.



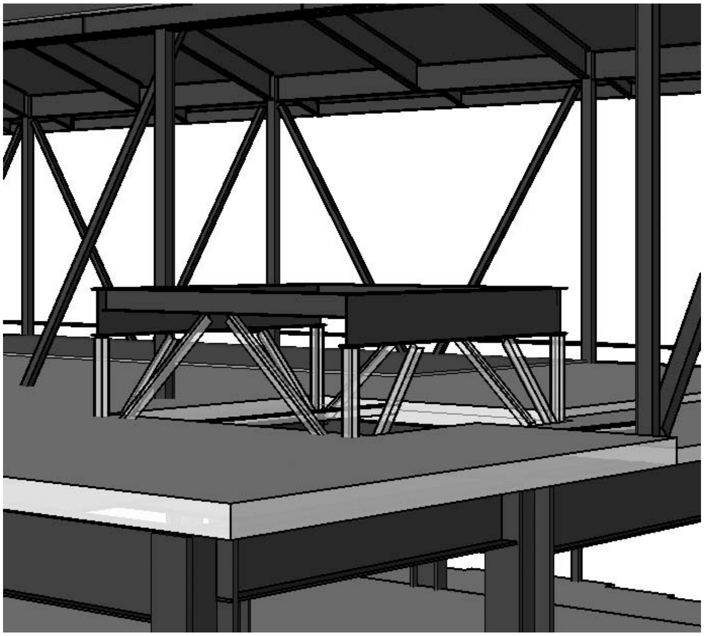
Our approach to typical details in this project was different than most other Revit Structure projects. Rather than insert stock typical details and adjust them, we derived most of those details in this project from the model. A good example is the foundation details. We created one cross section through the foundation area showing a wall footing, tie beam, and column isolated footing from the model. The detail displays the exterior wall detailing as well, and shows how it locks into the first floor slab.

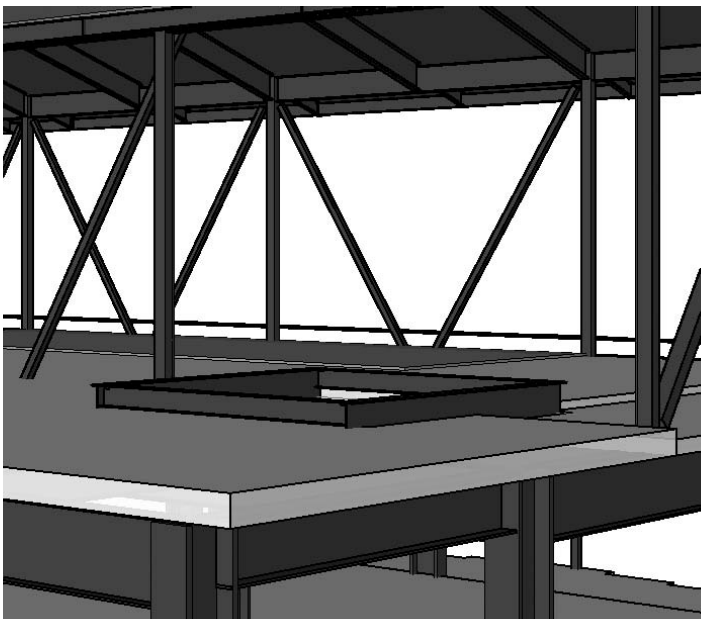


That detail represented most of the foundation area detailing. So rather than having to adapt a stock detail, we deemed this approach more efficient and direct.



Another interesting problem was a change that occurred during construction. Two mechanical equipment platforms on the roof were found to be too tall after they were erected. On one platform we decided to remove the posts and diagonal braces and bring the platform framing down onto the roof.





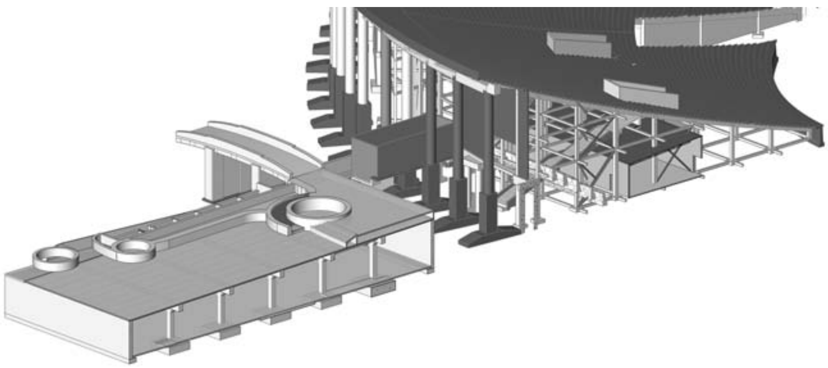
We made the effort much simpler by creating two design options and using phasing. One option represented the existing condition. The members to be removed were designated to be demolished and could be shown as hidden lines in the elevations. The other design option represented the new finished condition. We could then enable the two options as needed in the various views and document them to show the existing and new conditions.

This was not a huge project in terms of square feet, but as you see it had many conditions that had to be modeled in order to achieve a model that had sufficient integrity. Not every project is an 80-story high rise or an airport terminal. For your average structural firm, Revit Structure will do just fine in modeling structures. The mix of new and existing portions as well as the mix of steel, concrete, and wood can be just as demanding to design and model as that new fancy high rise.

If the boss asks if it wouldn't be easier to just do it in AutoCAD, your answer should be, "No way! Let me do it in Revit Structure!" Don't look back.

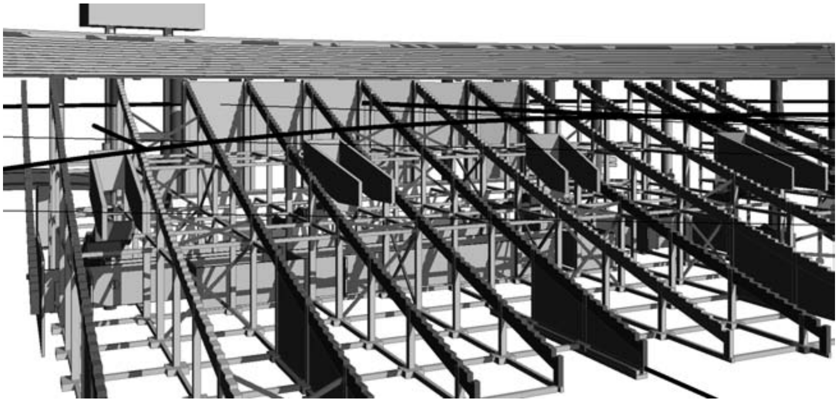
The Historic Rose Bowl Locker Room and Media Center Project

This is another high-profile project with a demanding schedule. The historic Rose Bowl was built in the early 1920s in the city of Pasadena, California. The UCLA football team is now one of their main tenants. A bigger and better locker room was required along with a new media center. The team was anxious to move into their new facility.



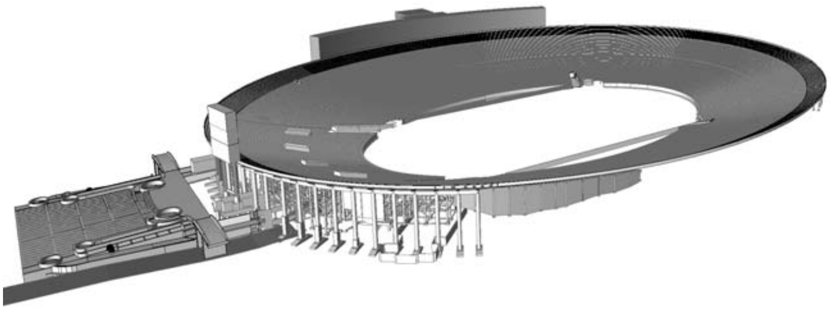
The Rose Bowl consists of a set of structures added over the years. As originally built, the Rose Bowl was a horseshoe shape, open at the south end. The south end was closed in the late 1920s and is the only part that is not sitting on grade. It is supported by concrete columns, braces, and beams.

In the initial design, the locker room was going to be under the south end of the bowl. But after development, that design was abandoned because it was too difficult to accomplish due to the nature of the existing structure. The south end of the Rose Bowl is so full of concrete columns, beams, and braces that it looks more like a spaghetti bowl!



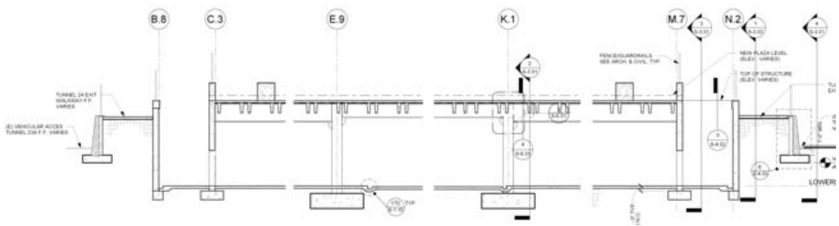
Then began the next design effort: to locate the locker rooms just outside of the stadium under the parking lot, with connecting tunnels onto the playing field. The media center remained tucked under the south end as a one-story structure.

The structural engineer on this project was contracted to provide all the engineering services required for the Rose Bowl. We decided that the entire existing historic bowl would be modeled. That sounds like a huge undertaking, and it was complex but doable. By centralizing the work into one ongoing model, we could divide the cost between successive contracts. The more complete the model, the better resource it became for facility management uses. This then was a real BIM solution.

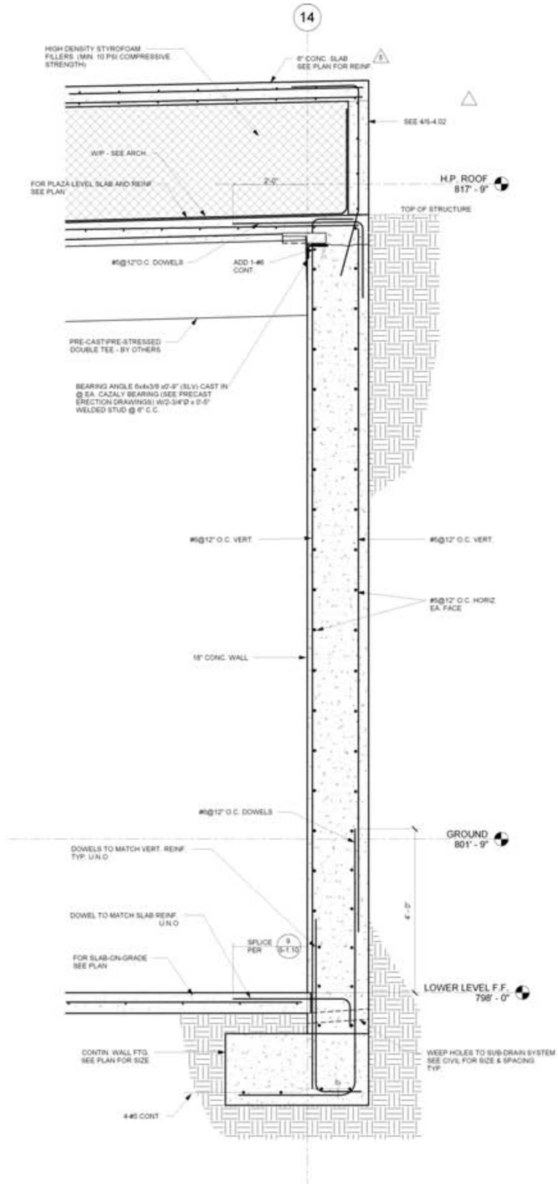


The bowl shape could be created in a straightforward way by using a solid sweep to create the basic shape. We drew the profile and then swept 360 degrees to create the basic bowl shape. Most of the Rose bowl structure is sitting on the ground. Fascinating old photographs are available showing mule-drawn wagons in the 1920s as they removed dirt from the site.

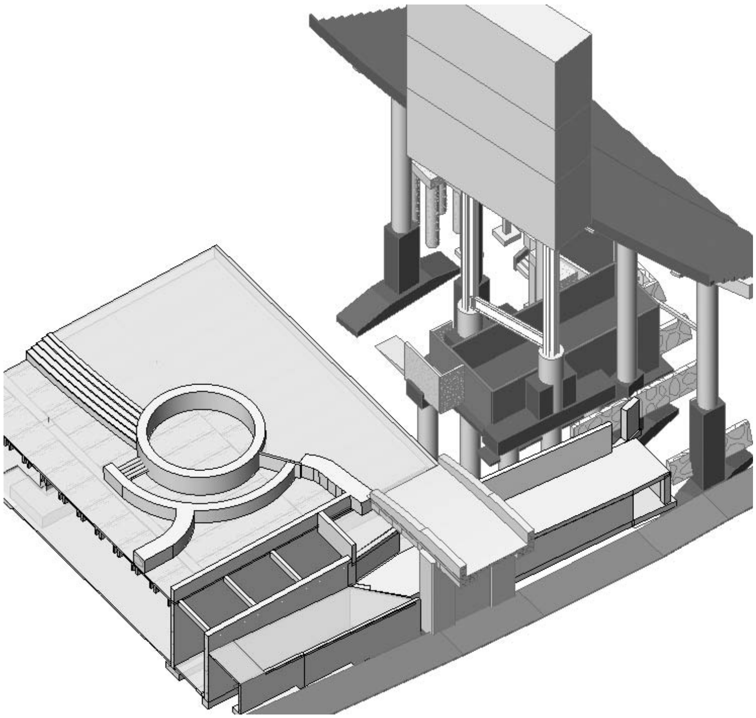
The new locker room scheme uses precast columns and girders. The roof is made of precast concrete double-T planks. Walls are cast-in-place concrete.



The Wall of Champions was another important design consideration in that we had to maintain accessibility. It showcases Rose Bowl history and is located on the south end. The precast concrete roof planks over the locker rooms are covered with about a foot of soil, with paving above that. We added elaborate concrete landscaping walls, with new stairs and a second slab-on-Styrofoam leading up to the Wall of Champions. All this was accomplished in Revit Structure with very little 2D work required.



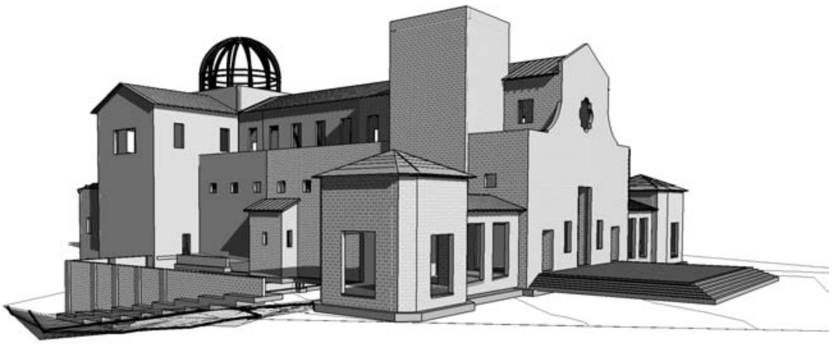
One of the main design problems with the new scheme involved constructing the tunnel while avoiding the many large foundations supporting the existing bowl and the large stadium sign. Ensuring the integrity and accuracy of the model became a crucial task that required successive site surveys. We created walkthroughs through the tunnels, and it became quite simple to spot any intrusion by the existing foundations. The tunnel construction also used design options. Options were required because the contractor wanted to explore the possibility of using masonry walls instead of concrete.



In the end, the second design scheme in the parking lot was adopted. We completed the design and construction work in record time with few RFIs, and now the team is ready to move in.

A Church Project

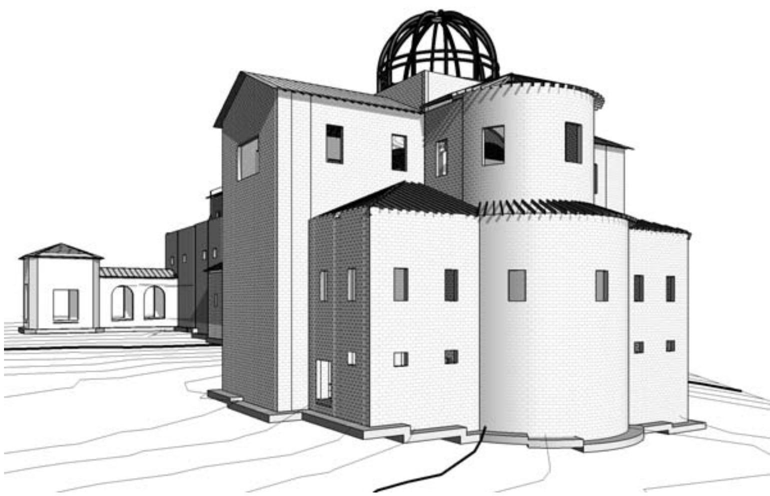
Religious structures come in many forms. In this case the project is a Roman Catholic church whose design is inspired by historic churches of California as well as Catholic tradition. As such, it has a nave, a large sanctuary, and a transept forming a traditional cross with a large dome over its altar. The Lady of the Most Blessed Trinity Chapel at St. Thomas Aquinas College is situated on a prominent site so that it is visible to all those passing through this beautiful Southern California Valley north of Los Angeles.



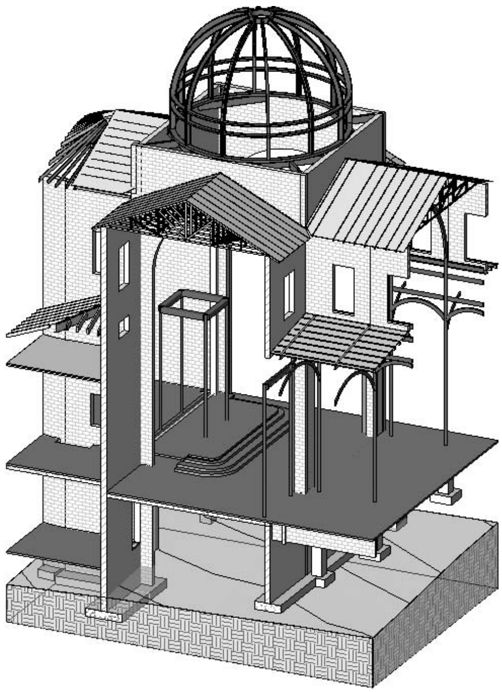
Because the church is in a valley, though, the site had a steep slope. One of the toughest design questions we faced was whether to in-fill the site and construct the church floor on grade or construct a partially supported floor. Which way was the most economical? Modeling the structure helped in that determination. Using the topography features in Revit Structure, we were able to import the 2D CAD site survey and construct a 3D site plan. It took a few hours to add contour points over the 2D contour lines and create the 3D topography. With that in place, we could cut sections through the model and see exactly where the existing hillside was located, which was a great aid while we were figuring how to step the foundation down the slope. In the end, the design decision was to use a partially supported floor.

We also decided early in the design process that the church steeple would not be designed by the structural engineer, but instead would be separately designed and built by others. As you see, it does not appear in the model views.

The church walls are made of masonry with a concrete foundation. The circular walls in the back proved difficult to create, especially since they were modeled in an earlier version of Revit Structure. Placing openings in the curved walls and adding the rounded roof framing was a real chore. New versions of Revit Structure make it easier to accomplish these tasks.



The altar area was the most demanding to model and engineer, due to the confluence of elements there. The dome had to be supported with arched walls below. The altar area was stepped, and several wood roof areas converged. We abstracted a partial model of that area, which became very useful in assessing the integrity of our design. We modeled the dome structural elements using an in-place family using a solid sweep to create the curved vertical steel members, then arraying them about the circle.



Most of the roofs in the sanctuary and transept areas were made from wood trusses. We created the trusses in Revit Structure using a simple solid extrusion, grouped, and then arrayed as necessary. We did this step before the truss wizard was available. The lower roof portions were constructed of lumber members framing into light steel framing members with nailers on top. Curved steel tube framing was required throughout the sanctuary area in order to support the architectural finishes.

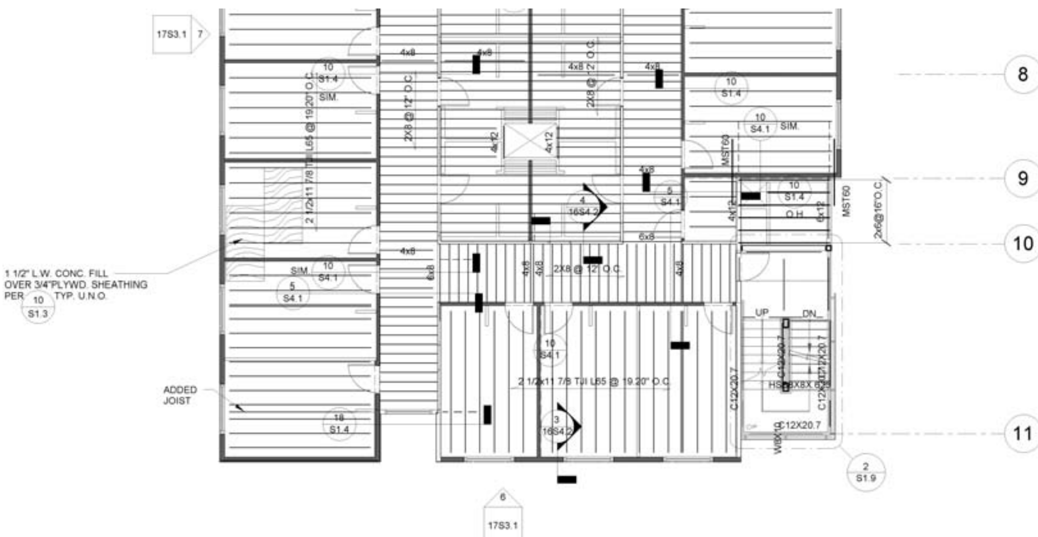
A Student Housing Project

Wood materials are used extensively in many parts of the United States and Canada. With the skyrocketing price of concrete and steel due to worldwide demand, we can expect to see more structures built of lumber products in the future, a renewable resource in this green era. But modeling in wood has its own challenges. There are just so many pieces of wood to contend with that you must be very careful in how you approach the virtual model.

This student housing project uses wood and light steel framing. Wood shear walls form the main seismic resisting system. There are two four-story structures for housing and a one-story support building.

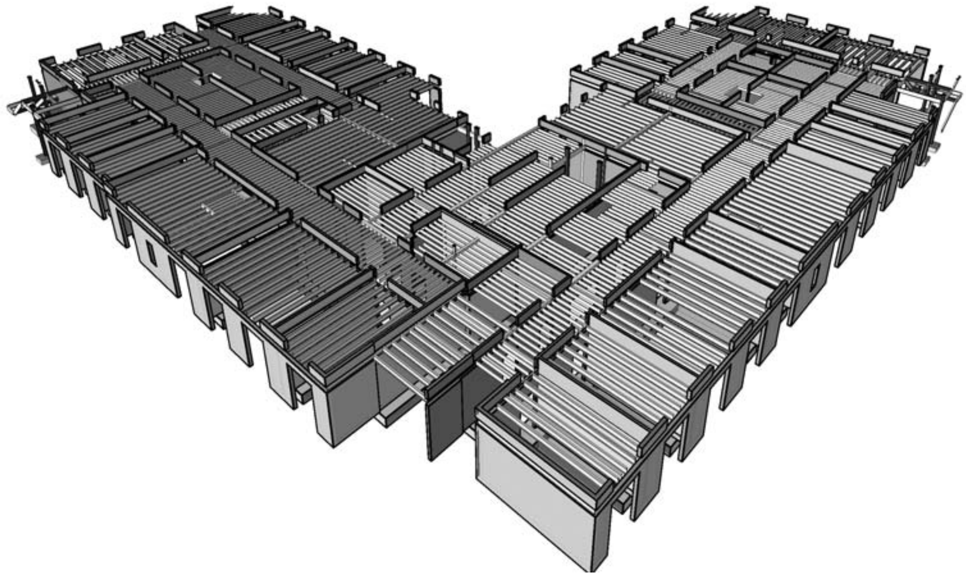
Shear Walls

The walls of the structures did not contain individual studs. For the purpose of composing construction documents, that was not necessary. As you saw in Chapter 19, in the shear wall definitions the plywood sheathing was added as a vertical layer. The top and bottom plates were inserted as sweeps with embedded detail components. That way, when a section was cut the wood sheathing and plates would automatically display. We created interior and exterior shear wall and bearing wall family types to give us greater control over the display and addition of curbs at the ground level. A 6″ curb could be added directly into the family. In plan view we created separate plans for the shear walls and the floor framing. This way, the view was much less cluttered. We used annotated symbols to indicate hold-down anchor locations as well as shear wall types and extents.



Floor Framing

Floor framing consisted of wood I-joists in the room areas and lumber in the shorter corridor areas. As opposed to the walls, all floor framing members were modeled. Most floor framing was created using the Beam System with a Maximum Spacing setting of 16″ center to center.

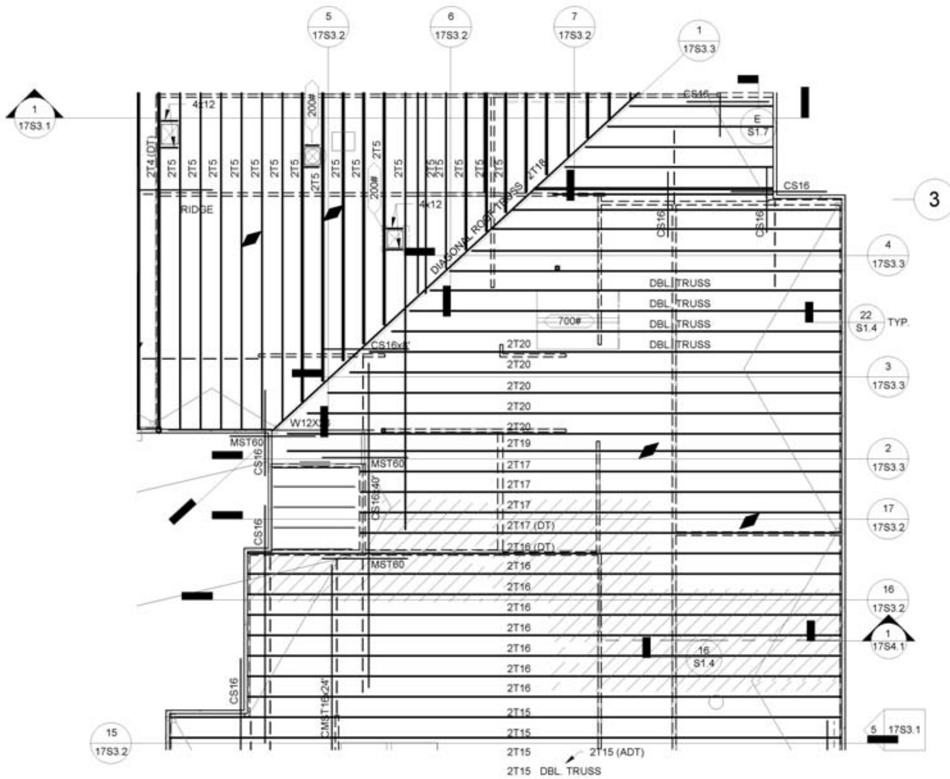


Roof Trusses

The roof trusses on this project were very demanding. The trusses spanned across the building and were supported on the shear walls in the corridors below. The centers of the truss were all along the same ridge line. But the extents of the trusses varied depending on the roof footprint at any one location. For a shorter truss, the ends were simply clipped and a vertical member inserted.

The truss spacing was at two feet center to center so there were a tremendous amount of different types, each with its own length. So how could we model those? And could they be edited easily if necessary during the design process?

The answer to the problem was to construct a truss family. We created the truss family in elevation as a solid extrusion. On each end we added a solid void with an instance label. That way, we could insert one into the roof and then adjust the voids on each end to shorten the length as needed. Then that could be arrayed over the area the truss length represented.



As was said earlier, having to deal with so many pieces of timber can be frustrating. Editing can be difficult, so plan well.

Summary

Hopefully we have shown you that all types of projects and structural systems can be accomplished by using Revit Structure. It may not always be crystal clear how these project types or structural systems should be modeled, but honestly, two projects that use the same structural system may be handled two completely different ways. Or the size or unique framing requirements may have to be modeled or documented in a certain way that the out-of-the-box program can't address. It is important for you to think about how you will approach each project before you start. Consider the downstream effects. Think about what you and the design team want to get out of the model, and model within those expectations. A lot of situations will pop up midway through a project, but as you overcome the small roadblocks and figure out ways to move through them, you will be able to carry your solutions into future projects.

Index

Index

Note to the Reader: Throughout this index **boldfaced** page numbers indicate primary discussions of a topic. *Italicized* page numbers indicate illustrations.

B

C

E

G

H

I

J

K

L

O

T

U

V

W

Y

Structure Your Designs the BIM Way

Whether you're a structural engineer, designer, or draftsperson making the move to BIM, you'll dramatically increase productivity and become a Revit Structure expert with the techniques, workflows, and previously undocumented tips and tricks in this thorough reference and tutorial.

The expert authors combine their years of Revit and engineering experience to offer you a solid foundation in Revit Structure concepts, before moving to the structural modeling of such elements as walls, frames, and slabs. You'll learn advanced family creation, standards development, and model documentation and presentation–as well as such crucial topics as annotation, dimensioning, exporting data, 3D visualization, templates, publishing, collaboration, and more. With detailed tutorials and real-world solutions you can implement right away, this is the perfect resource for anyone who wants to master Revit Structure 2009.

COVERAGE INCLUDES:

- Understanding how parametric modeling frees you to focus on your designs
- Exploring the broad array of tools in Revit Structure's user interface
- Viewing and referencing plan, section, elevation, and model views–simultaneously–while you design
- Applying scheduling, annotation, and dimensioning info as you go, for speedier workflows
- Creating and presenting 3D visualizations that bring your designs to life
- Using analytical models to test and simulate, then refine your designs
- Adding custom content and building your own component libraries

Increase Your Productivity, Speed, and Accuracy with Revit Structure

Create Dynamic Structural Models Using Best Practices

Design, Draw, Detail, and Draft in a Virtual World

Compare Your Designs to Those of Revit Professionals on the Companion Website

View Striking Revit Structure Examples in the Full-Color Insert

ABOUT THE AUTHORS

Thomas S. Weir is the Director of BIM and CAD Operations at Brandow & Johnston in Los Angeles. He is President and founder of the L.A. Revit Users Group, moderates the Autodesk User Group International (AUGI) Revit Structural forum, teaches Revit Structure at Autodesk University, and wrote the first Autodesk Official Training Courseware for Revit Structure. **Eric Wing** has been in the AEC field for fifteen years and has been managing, teaching, and presenting Autodesk applications for ten of them. Eric is AUGI's Autodesk Training Program Director and is also a columnist for AUGI's *HotNews* monthly newsletter as well as for *AUGIWorld* magazine. **Jamie D. Richardson** is an Associate and a CAD/BIM manager for Ericksen Roed & Associates located in the Twin Cities area. He speaks at Autodesk University, is active in his local Revit User Group, and mentors students at local technical colleges. **David J. Harrington,** former president and former board member of AUGI, is currently a structural designer for Walter P. Moore. He also writes for industry publications, is the technical editor for *AUGIWorld* magazine, teaches at Autodesk University, and maintains a popular blog (caddhelp.blogspot.com).

www.sybex.com
www.sybex.com/go/masteringrevitstructure2009

ISBN 978-0-470-38440-4

56999

9 780470 384404

Sybex®
An Imprint of
WILEY

Autodesk
Authorized Publisher

Autodesk
Authorized Author

CATEGORY
COMPUTERS/CAD-CAM

$69.99 US
$76.99 CAN